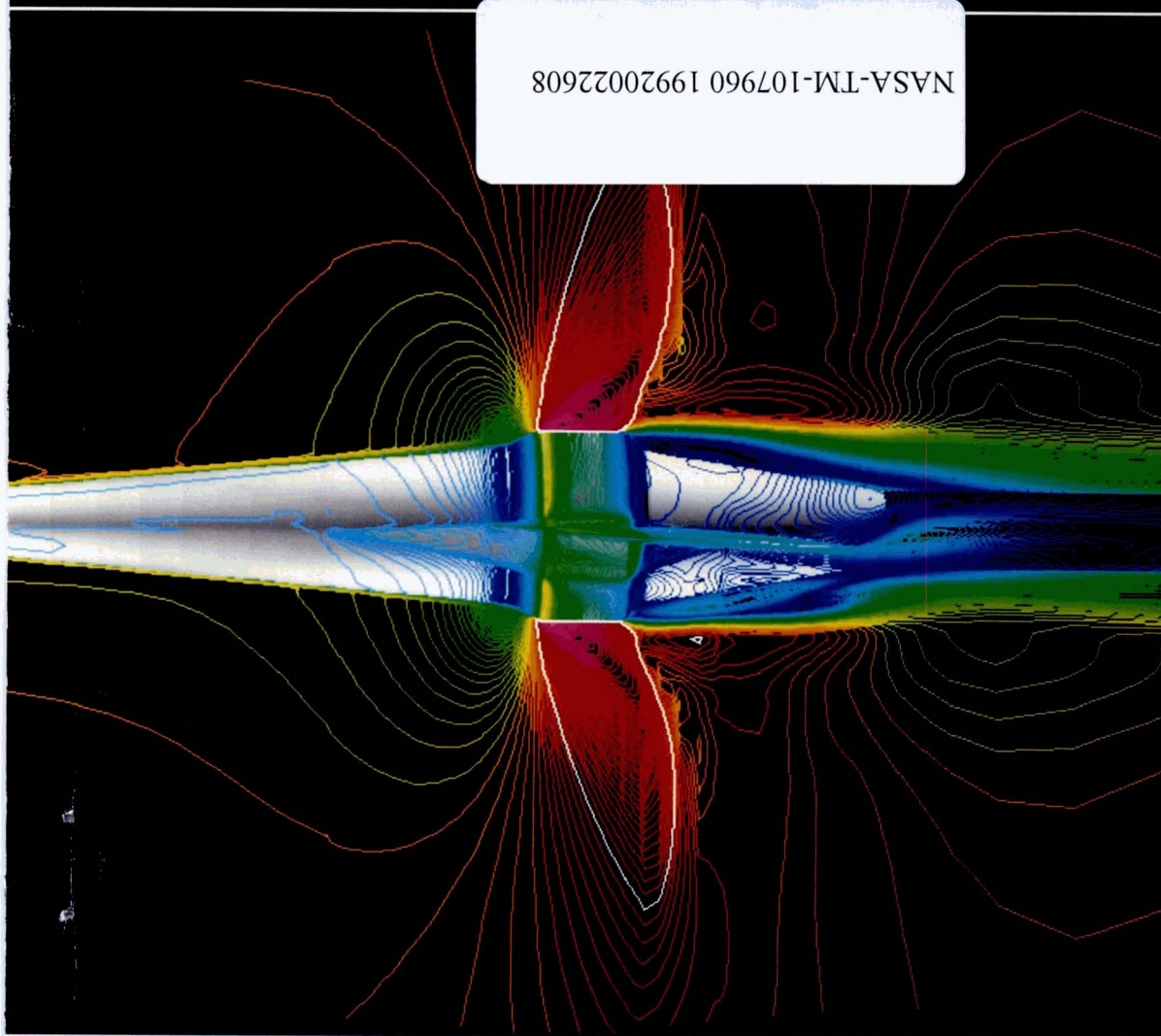


NAS

NUMERICAL
AERODYNAMIC
SIMULATION
PROGRAM

Technical Summaries

NASA-TM-107960 19920022608



MARCH 1989 – FEBRUARY 1990

The use of color graphics is a powerful and flexible visualization tool for the science of computational fluid dynamics. Without color graphics, the millions of numbers which form the supercomputer solutions to the studies contained in this report and on the cover would be virtually incomprehensible.

NAS NUMERICAL AERODYNAMIC SIMULATION PROGRAM

Technical Summaries

MARCH 1989 – FEBRUARY 1990

(NASA-TM-107960) NAS (NUMERICAL
AERODYNAMIC SIMULATION PROGRAM)
TECHNICAL SUMMARIES, MARCH 1989 –
FEBRUARY 1990 (NASA) 221 p

N92-31852

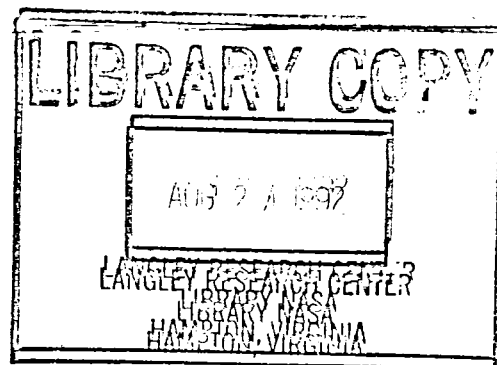
Unclas

G3/99 0114398



National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035-1000



N92-31852#
114998-1W

Preface

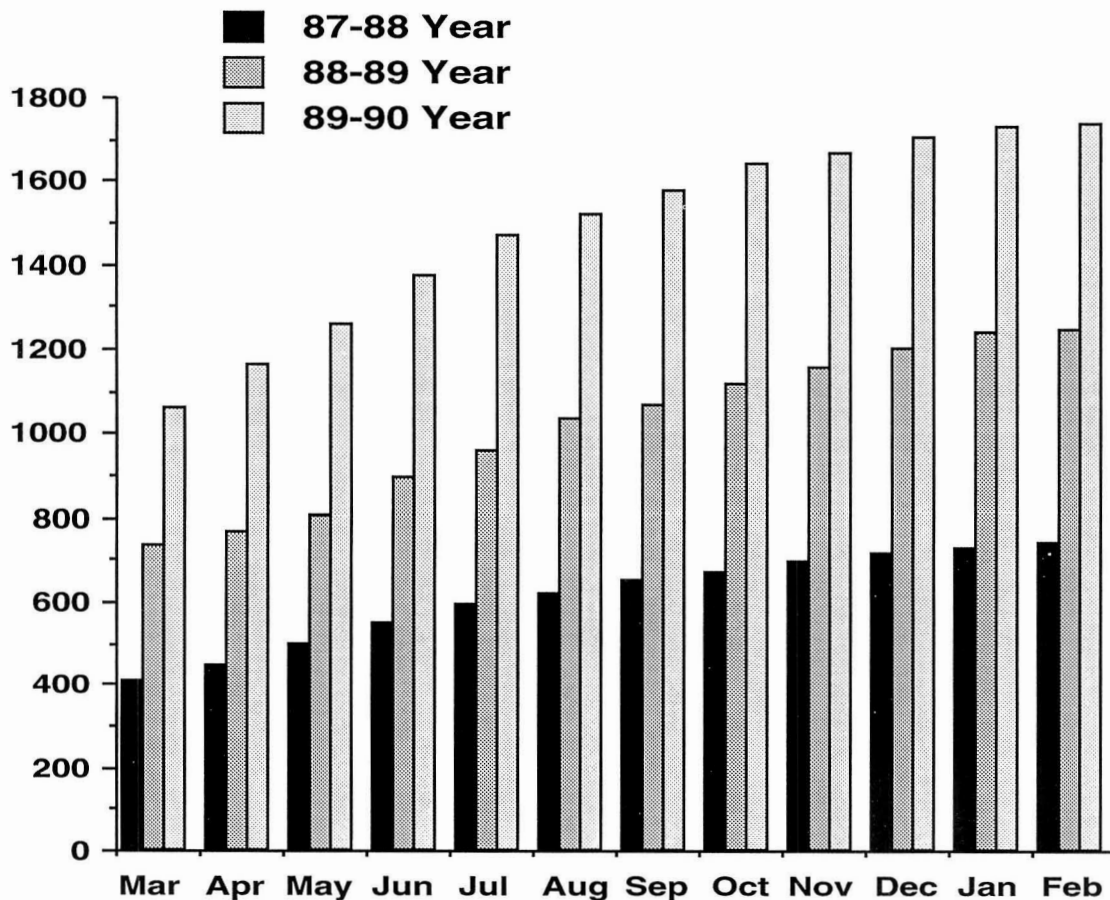
NASA created the Numerical Aerodynamic Simulation (NAS) Program to focus resources on solving critical problems in aerospace and related disciplines by utilizing the power of the most advanced supercomputers available. The NAS Program not only provides scientists with the computing power they need to solve today's most demanding computational fluid dynamics problems, it also benefits other supercomputer centers, both in Government and industry, by serving as a pathfinder in integrating leading-edge supercomputing technologies.

This report contains selected scientific results from the NAS Program's third year of operation, designated as the 1989-90 Operational Year. The year began March 1, 1989, and ended March 4, 1990. During this year, the scientific community was given access to a Cray-2 and a Cray Y-MP supercomputer. The Cray-2, the NAS Program's first-generation supercomputer, has four processors, 256 megawords of central memory, and a total sustained speed of 250 million floating-point operations per second (250 MFLOPS).

The second-generation supercomputer, the Cray Y-MP, has eight processors producing a total sustained processing speed of 1 billion floating-point operations per second (1 GFLOPS). The Cray Y-MP had 32 megawords of central memory for most of the year and was upgraded to 128 megawords in February 1990. The Cray Y-MP also has 256 megawords of SSD memory. The Mass Storage Subsystem, which provides the primary file storage for the user community, improved significantly during this operational year. In addition to 260 gigabytes of online disk, two Storage Technology tape robots were installed to provide 2.4 terabytes of archival data storage.

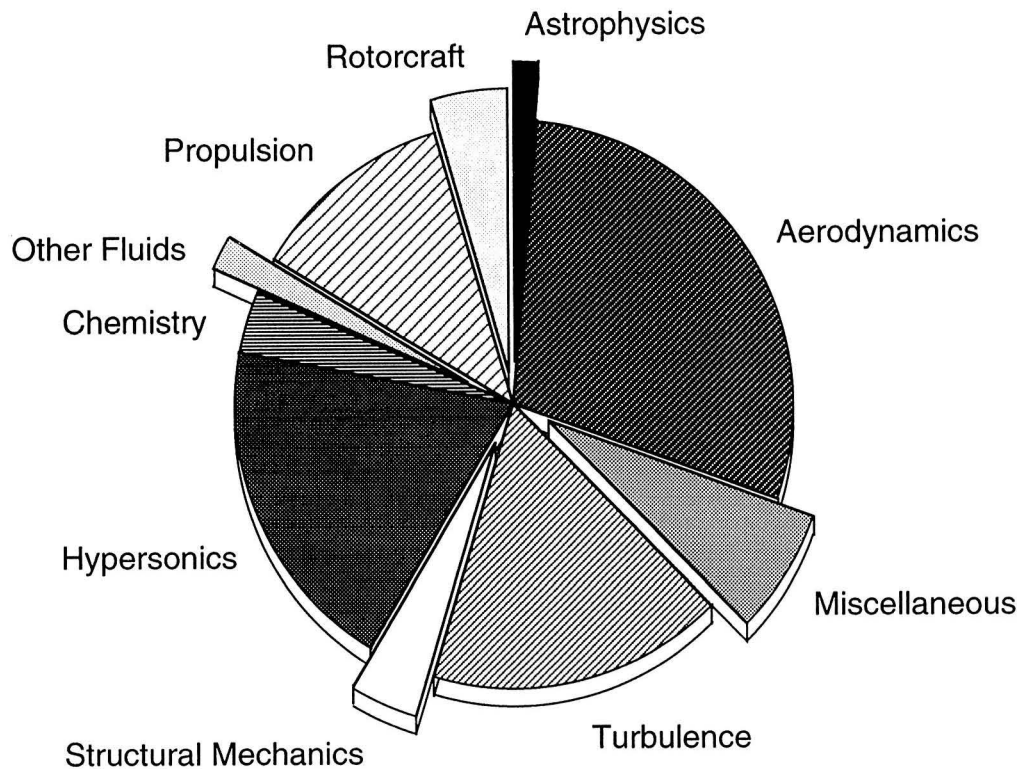
The size of the scientific user community continued to increase. The total number of users for the first year was 740, and the total for the second year increased to 1246. By the end of this year, the number of users reached 1733. The users were from 140 sites throughout the United States and represented NASA, the Department of Defense, and other Government agencies; private industry; and universities.

NAS Users



A total of 459 individual research projects were accepted to use the NAS resources during the 1988-89 Operational

Year. Of these projects, 89% were aeronautics oriented, as shown in the following chart.



At the end of each operational year, a NAS Technical Summaries Report is published to provide an overview of the significant scientific results for that year. To order either

additional copies of the report for this operational year or to order reports from previous years, please address your request to:

NAS Documentation Center
NASA Ames Research Center
Mail Stop 258-6
Moffett Field, CA 94035-1000
(415) 604-4632

Table of Contents

| Principal Investigator | NAS Summary | Page |
|-------------------------------|---|-------------|
| David Adamec | <i>Large-Scale Ocean Modeling</i> Co-investigators: Michele M. Rienecker and Antonio J. Busalacchi NASA Goddard Space Flight Center | 1 |
| Ramesh K. Agarwal | <i>Helicopter Fuselage and Rotor Flow-Field Calculations in Hover and Forward Flight</i> Co-investigator: Jerry E. Deese McDonnell Douglas Research Laboratories/NASA Ames Research Center | 2 |
| Ramesh K. Agarwal | <i>Numerical Solution of the Reynolds-Averaged Navier-Stokes Equations for Flow About an Almost Compete Aircraft</i> Co-investigator: Jerry E. Deese McDonnell Douglas Research Laboratories/NASA Ames Research Center | 3 |
| Shreekant Agrawal | <i>Separated and Vortical Flow-Field Analysis on Fighter Wing Configurations</i> Co-investigators: Tom A. Kinard and Brian A. Robinson McDonnell Aircraft Company | 4 |
| Donald Baganoff | <i>Discrete Particle Simulation of Compressible Flow</i> Co-investigators: William J. Feiereisen, Jeffrey McDonald, Brian Haas, Michael Woronowicz, Michael Fallavollita, and Leo Dagum Stanford University/NASA Ames Research Center | 5 |
| R. Balasubramanian | <i>Direct Simulation of Large-Eddy Breakup Devices</i> Spectrex Inc. | 6 |
| John T. Batina | <i>Validation of a Three-Dimensional Flux-Split Euler Algorithm for Unstructured Grids</i> Co-investigators: Elizabeth M. Lee, Russ D. Rausch, and William L. Kleb NASA Langley Research Center | 7 |
| William W. Bower | <i>Calculations of Unsteady, Supersonic-Jet Flow Fields and Acoustics</i> Co-investigators: Robert E. Childs and Gerald E. Chmielewski McDonnell Douglas Research Laboratories/Nielsen Engineering & Research, Inc. | 8 |
| Stephen H. Brecht | <i>Hybrid Simulations of the Global Venus/Solar Wind Interaction</i> Berkeley Research Associates | 9 |
| Edwin B. Brewer | <i>Direct Simulation of Aerothermal Loads for the Aeroassist Flight Experiment</i> NASA Marshall Space Flight Center | 10 |
| Adam C. Bricker | <i>Hypersonic Air-Breathing Missile Aero/Propulsion Integration</i> Co-investigators: Brad K. Bergman, Kurt Abrahamson, and Nancy Wilkins General Dynamics, Convair Division | 11 |
| Jeffrey C. Buell | <i>Three-Dimensional Shear Layers and Wall-Bounded Compressible Turbulence</i> NASA Ames Research Center | 12 |
| Pieter G. Buning | <i>Space Shuttle Flow Field</i> Co-investigators: J. L. Steger, R. L. Meakin, F. W. Martin, I.-T. Chiu, Y. M. Rizk, M. Yarrow, and S. Obayashi NASA Ames Research Center | 13 |
| Geoffrey Butler | <i>Two-Body Hypersonic Separation Analysis</i> Co-investigators: Brian Nguyen, Eric Sharnhorst, and David King General Dynamics, Convair Division | 14 |
| Alan B. Cain | <i>Receptivity, Eigenfunction Modeling, and Simulation of a Wall-Bounded Flow</i> Co-investigators: William W. Bower and Nagi N. Mansour McDonnell Douglas Research Laboratories/NASA Ames Research Center | 15 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|--|-------------|
| Steven C. Caruso | <i>An Unstructured Triangular-Mesh/Navier-Stokes Method for Computing the Aerodynamics of Aircraft with Ice Accretion</i> Nielsen Engineering & Research, Inc. | 16 |
| Sukumar R. Chakravarthy | <i>Fuel/Air Turbulent Combustion in a Three-Dimensional Combustor Geometry in Supersonic Flow</i> Co-investigators: Uriel C. Goldberg and Sampath Palaniswamy Rockwell International Science Center | 17 |
| R. Rexford Chamberlain | <i>Three-Dimensional Calculations of Multiple Control-Jet Interactions for Defense Interceptors</i> Lockheed Missiles and Space Company, Inc. | 18 |
| Dean R. Chapman | <i>Continuum Computational Fluid Dynamics for Hypersonic High-Altitude Flight</i> Co-investigator: Robert W. MacCormack Stanford University | 19 |
| Denny S. Chaussee | <i>Hypersonic Flow Past Generic Lifting Bodies</i> Co-investigators: Scott L. Lawrence, Thomas A. Edwards, and Bradford Bennett NASA Ames Research Center | 20 |
| Lee T. Chen | <i>Calculation of Transonic and Supersonic Flows Using an Interactive Scheme Based on Euler and Boundary-Layer Equations</i> Co-investigators: Minh H. Bui and Hsun H. Chen Douglas Aircraft Company | 21 |
| W. J. Chyu | <i>Airframe/Inlet Aerodynamics</i> NASA Ames Research Center | 22 |
| William J. Coirier | <i>Three-Dimensional Shock/Boundary-Layer Interactions in Hypersonic Inlets</i> NASA Lewis Research Center | 23 |
| William B. Compton III | <i>Transonic Navier-Stokes Solutions of Three-Dimensional Afterbody Flows</i> Co-investigator: Khaled Sayed Abdol-Hamid NASA Langley Research Center/Analytical Services and Materials, Inc. | 24 |
| Raymond R. Cosner | <i>Hornet 2000 Flow-Field Analysis</i> Co-investigators: Shreekanth Agrawal and Patrick J. Malloy McDonnell Aircraft Company | 25 |
| Russell B. Dahlburg | <i>Dynamical Modeling of the Solar Atmosphere</i> Co-investigators: Jill P. Dahlburg, John T. Mariska, and J. M. Picone Naval Research Laboratory | 26 |
| Roger L. Davis | <i>The Impact of Hot-Streak Migration on Turbine Heat Transfer</i> Co-investigators: Daniel J. Dorney and Diane M. Rodimon United Technologies Research Center | 27 |
| Bill Davy | <i>Three-Dimensional Aeroassist Flight Experiment Flow Simulations</i> Co-investigators: Grant Palmer, Dinesh Prabhu, and Ellis Whiting NASA Ames Research Center | 28 |
| Pramote Dechaumphai | <i>Integrated Fluid-Thermal-Structural Analyzer Demonstrates Heat-Transfer/Deformation Coupling</i> NASA Langley Research Center | 29 |
| Nabeel A. O. Demerdash | <i>Finite-Element Computation of Three-Dimensional Magnetic Fields for Design Optimization of Generators for Space Station Solar Dynamic Power</i> Co-investigator: R. Wang Clarkson University | 30 |
| Robert DeParvine | <i>High-Angle-of-Attack Inlet Analysis and Design Using Computational Fluid Dynamics Methods</i> Co-investigator: Ben Smith General Dynamics, Fort Worth Division | 31 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|--|-------------|
| Robert DeParvine | <i>Validation of a Computational Fluid Dynamics Code for High-Speed-Inlet Design</i> Co-investigator: Ben Smith General Dynamics, Fort Worth Division | 32 |
| Dan F. Dominik | <i>Advanced Solid Rocket Motor Study</i> Co-investigators: Vedat Akdag, Shmuel Ben-Shmuel, William Riba, Robert Williams, Cheng L. Chen, S. Ramakrishnan, K. Ragagopal, and S. Chakravathy Rockwell International, Space Transportation Systems Division and Science Center | 33 |
| J. Philip Drummond | <i>Simulation of Supersonic Chemically Reacting Flow Fields</i> Co-investigators: Mark H. Carpenter, Peyman Givi, Johnny R. Narayan, David W. Riggins, Balu Sekar, and Jeffrey A. White NASA Langley Research Center | 34 |
| George S. Dulikravich | <i>Acceleration of Iterative Algorithms for Euler and Navier-Stokes Equations</i> Co-investigator: Seungsoo Lee The Pennsylvania State University | 35 |
| John K. Eaton | <i>The Interaction of Particles with Homogeneous Turbulence</i> Co-investigator: Kyle D. Squires Stanford University | 36 |
| Thomas A. Edwards | <i>Hypersonic Chemically Reacting Flow over Blended Wing-Body Vehicles</i> Co-investigators: Jolen Flores, Uwe Jettmar, and Greg Molvik NASA Ames Research Center | 37 |
| Thomas A. Edwards | <i>Numerical Simulation of Hydrogen-Air Combustion in Hypersonic-Vehicle Propulsion Systems</i> Co-investigators: Jolen Flores and Uwe Jettmar NASA Ames Research Center | 38 |
| T. Alan Egolf | <i>Advanced CFD Codes for Rotary-Wing Airloads and Performance Prediction</i> Co-investigator: Brian E. Wake United Technologies Research Center | 39 |
| S. E. Elghobashi | <i>Direct Simulation of Turbulent Reacting Flows</i> Co-investigator: K. K. Nomura University of California, Irvine | 40 |
| S. E. Elghobashi | <i>Dispersion of Solid and Fluid Particles in Turbulent Homogeneous Flows With and Without Turbulence Modulation</i> Co-investigator: G. C. Truesdell University of California, Irvine | 41 |
| N. M. El-Hady | <i>Control of the Flow Field about an Airfoil Section by Localized Surface Heating</i> Co-investigator: L. Maestrello NASA Langley Research Center/Analytical Services and Materials, Inc. | 42 |
| John J. Erhart | <i>CFD Analysis of the Flow Paths of Spherical Convergent Flap Nozzles</i> Co-investigators: Saadat A. Syed and Eric W. King United Technologies, Pratt & Whitney | 43 |
| G. Erlebacher | <i>Compressible Turbulence</i> Co-investigators: S. Sarkar, C. Shu, and T. A. Zang NASA Langley Research Center/ICASE | 44 |
| Fort F. Felker | <i>Three-Dimensional Viscous Drag Prediction for Rotor Blades</i> Co-investigator: Ching S. Chen NASA Ames Research Center | 45 |
| Jolen Flores | <i>Compressible Navier-Stokes Code Development</i> Co-investigators: Steve Ryan and Tom Edwards NASA Ames Research Center | 46 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|--|-------------|
| Arthur J. Freeman | <i>Electronic-Structure Study of High Superconductors</i> Co-investigators: A. Continenza, S. Massidda, and J. Yu Northwestern University | 47 |
| K.-Y. Fung | <i>Computational Studies of Compressibility Effects on Dynamic Stall</i> University of Arizona | 48 |
| Javier A. Garriz | <i>Validation of 3-D Wind Tunnel Wall Interference Assessment/Correction Codes</i> NASA Langley Research Center | 49 |
| Gary A. Glatzmaier | <i>Numerical Simulations of Deep Global Convection in Jupiter</i> Los Alamos National Laboratory | 50 |
| Peter A. Gnoffo | <i>Hypersonic Flows in Chemical and Thermal Nonequilibrium</i> NASA Langley Research Center | 51 |
| Robert A. Golub | <i>Flow-Field Simulation Around Different Rotor Planforms</i> Co-investigator: Forooz F. Badavi NASA Langley Research Center/Lockheed Engineering and Sciences Company | 52 |
| Aga M. Goodsell | <i>A Supersonic Computational Fluid Dynamic Analysis of a Generic Fighter</i> NASA Ames Research Center | 53 |
| Fernando F. Grinstein | <i>Simulation of Compressible, Spatially Evolving, Reactive Planar Shear Flows</i> Naval Research Laboratory | 54 |
| W. L. Grose | <i>Three-Dimensional Atmospheric Simulation Model</i> Co-investigators: W. T. Blackshear, R. W. Turner, and R. S. Eckman NASA Langley Research Center | 55 |
| G. P. Guruswamy | <i>Simulation of Fluid/Structural Interactions</i> Co-investigators: S. Obayashi, N. M. Chaderjian, and P. M. Goorjian NASA Ames Research Center | 56 |
| Karl E. Gustafson | <i>Vortex Dynamics in Aerodynamic Flows</i> Co-investigator: Robert R. Leben University of Colorado, Boulder | 57 |
| Kadosa Halasi | <i>Parameter Study in the Driven Cavity</i> Co-investigator: Karl E. Gustafson Kansas State University/University of Colorado, Boulder | 58 |
| Edward J. Hall | <i>Ducted Propfan Analysis</i> Allison Gas Turbine Division, General Motors Corporation | 59 |
| Richard L. Haller | <i>Evaluation of the CAP-TSD Computer Code Using an F-16 Study Case</i> Co-investigators: Kevin Penning and Lam-son Vinh General Dynamics, Fort Worth Division | 60 |
| Julius E. Harris | <i>Supersonic Laminar-Flow-Control Computational Fluid Dynamics</i> Co-investigators: Venkit Iyer and Robert E. Spall NASA Langley Research Center | 61 |
| Peter M. Hartwich | <i>General, Fast, Accurate Floating Shock Fitting Procedure for Unadapted Meshes</i> ViGYAN, Inc./NASA Langley Research Center | 62 |
| Peter M. Hartwich | <i>Navier-Stokes Solutions for Vortical Flows over a Forebody</i> ViGYAN, Inc./NASA Langley Research Center | 63 |
| H. A. Hassan | <i>Monte Carlo Simulation of Hypersonic Flows</i> Co-investigators: David P. Olynick and Jeff C. Taylor North Carolina State University | 64 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|---|-------------|
| H. A. Hassan | <i>Turbulence Modeling of Separated Flows</i> Co-investigators: Robert A. Mitcheltree and Richard L. Gaffney, Jr. North Carolina State University | 65 |
| H. A. Hassan | <i>Turbulent Supersonic Mixing Layers</i> Co-investigators: Dean R. Eklund, Steven H. Frankel, and Erick J. S. Gantt North Carolina State University | 66 |
| Henry J. Haussling | <i>Multiblock Solutions of the Navier-Stokes Equations</i> Co-investigators: Joseph J. Gorski and Roderick M. Coleman David Taylor Research Center | 67 |
| James D. Heidmann | <i>An Analysis of the Viscous Flow through a Compact Radial Turbine by the Average-Passage Approach</i> NASA Lewis Research Center | 68 |
| James C. Hill | <i>Direct Simulation of Turbulent Reacting Flows</i> Co-investigators: Andy D. Leonard and Dana G. Haugli Iowa State University | 69 |
| Scott D. Holland | <i>A Computational and Experimental Parametric Study of Three-Dimensional Side-Wall Compression Scramjet Inlets at Mach 10</i> Co-investigator: John N. Perkins North Carolina State University | 70 |
| C. C. Horstman | <i>Turbulence Modeling for Compressible/Hypersonic Flows</i> Co-investigators: J. R. Viegas, T. J. Coakley, and P. Rodi NASA Ames Research Center | 71 |
| C.-H. Hsu | <i>Prediction of Vortical Flows Using Incompressible Navier-Stokes Equations</i> ViGYAN, Inc./NASA Langley Research Center | 72 |
| Thomas T. Huang | <i>Turbulence Modeling for Submarine Flow Field Computation</i> Co-investigator: Yu-Tai Lee David Taylor Research Center | 73 |
| Gary W. Huband | <i>Numerical Simulation of an F-16A at Angle of Attack</i> Co-investigators: J. S. Shang and Michael J. Aftosmis Wright Research and Development Center | 74 |
| Ching-mao Hung | <i>Computation of Steady Three-Dimensional Separation</i> NASA Ames Research Center | 75 |
| Harley E. Hurlburt | <i>Eddy-Resolving Model of the Pacific Ocean</i> Co-investigators: Alan J. Wallcraft and Jimmy L. Mitchell Naval Oceanographic and Atmospheric Research Laboratory/JAYCOR | 76 |
| Danny Hwang | <i>Computational Advanced Propulsion Technology Research</i> Co-investigators: John Wolter, Joe Nenni, Mark Krein, and Dave Georgevich NASA Lewis Research Center | 77 |
| Steven H. Izen | <i>Numerical Inversion of a Limited-Data X-Ray Transform</i> Case Western Reserve University | 78 |
| Laurence R. Keefe | <i>Linking Attractor Geometry to Turbulence Physics</i> NASA Ames Research Center | 79 |
| John L. Klann | <i>Solar Dynamic Space Power-System Design and Performance Analysis</i> Analex Corporation/NASA Lewis Research Center | 80 |
| Ajay Kumar | <i>Three-Dimensional Shock/Shock Interactions on the Inlet Side Wall</i> Co-investigator: Dal J. Singh NASA Langley Research Center | 81 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|---|-------------|
| Geojoe Kuruvila | <i>Numerical Study of Fundamental Fluid Dynamics using Navier-Stokes Equations</i> NASA Langley Research Center | 82 |
| Dochan Kwak | <i>Incompressible Navier-Stokes Calculations for the Space Shuttle Main Engine</i> Co-investigators: Stuart Rogers, Seokkwan Yoon, Moshe Rosenfeld, and Cetin Kiris NASA Ames Research Center | 83 |
| H. T. Lai | <i>Computation of Single-Expansion-Ramp and Scramjet Nozzles</i> Sverdrup Technology, Inc. | 84 |
| C. C. Lee | <i>Numerical Investigation of NASP-Like Vehicles</i> Co-investigators: William Bower, Shawn Hagmeier, Scott Van Horn, Pat Vogel, and Darrell Weber McDonnell Douglas Corporation | 85 |
| S. J. Leib | <i>Nonlinear Instability of Shear Layers</i> Sverdrup Technology, Inc./NASA Lewis Research Center | 86 |
| S. J. Lin | <i>Computational Fluid Dynamics Analysis of Space Shuttle Main Engine Turbopump Rotor/Stator Flows in Two and Three Dimensions</i> Co-investigator: R. J. Yang Rockwell International, Rocketdyne Division | 87 |
| Rung T. Ling | <i>Application of Computational Fluid Dynamics Methods to Radar Cross Section Computations</i> Northrop Aircraft Division | 88 |
| C. H. Liu | <i>Computational Prediction and Control of Steady and Unsteady Asymmetric Vortex Flows around Cones</i> Co-investigator: Osama A. Kandil NASA Langley Research Center/Old Dominion University | 89 |
| James M. Luckring | <i>Transonic Navier-Stokes Solutions about a Complex High-Speed Accelerator Configuration</i> Co-investigators: Farhad Ghaffari, James L. Thomas, and Brent L. Bates NASA Langley Research Center | 90 |
| Gregory A. Lyzenga | <i>Earthquake-Cycle Stress Simulations</i> Co-investigators: Arthur Raefsky and Stephanie G. Mulligan Jet Propulsion Laboratory/California Institute of Technology | 91 |
| Edward C. Ma | <i>Numerical Investigation of a Space Shuttle Orbiter Contingency Abort</i> Co-investigator: Thomas C. Wey Lockheed Engineering and Sciences Company | 92 |
| M. G. Macaraeg | <i>Nonlinear Evolution of Supersonic Disturbances in Mixing Layers</i> NASA Langley Research Center | 93 |
| Robert W. MacCormack | <i>Simulation of Highly Ionized Flows</i> Co-investigator: Philippe Rostand Stanford University/Analatom, Inc. | 94 |
| Robert D. MacElroy | <i>Computer Simulation of Ion Transport across Membranes</i> Co-investigator: Andrew Pohorille NASA Ames Research Center | 95 |
| Michael D. Madson | <i>Transonic Analysis About the F-16A and Other Complex Configurations</i> Co-investigators: Alex C. Woo, Ralph L. Carmichael, F. T. Johnson, S. S. Samant, D. P. Young, R. G. Melvin, M. B. Bieterman, and J. E. Bussoletti NASA Ames Research Center/Boeing Advanced Systems | 96 |
| L. Maestrello | <i>Radiation and Control of the Acoustic Field Caused by a Wave Packet Evolving along a Concave-Convex Surface</i> Co-investigator: N. M. El-Hady NASA Langley Research Center/Analytical Services and Materials, Inc. | 97 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|--|-------------|
| Mujeeb R. Malik | <i>Transition in Hypersonic and Three-Dimensional Boundary Layers</i> Co-investigators: C.-L. Chang and R. E. Spall High Technology Corporation/NASA Langley Research Center | 98 |
| John Mangus | <i>Navier-Stokes Analysis of the Air Force Vortex Flap Model and the F/A-18</i> Co-investigators: Shreekant Agrawal, Robert Lowrie, and Brian Robinson Northrop Aircraft Division/McDonnell Aircraft Company | 99 |
| Nagi N. Mansour | <i>Direct Simulation of Compressible Turbulent Flows</i> Co-investigators: J. Chen and G. Blaisdell NASA Ames Research Center | 100 |
| Fred W. Martin, Jr. | <i>Space Shuttle Flow Field</i> Co-investigators: Joseph Steger, Pieter Buning, Steve Labbe, Ray Gomez, Phil Stuart, Jeff Slotnick, Steve Parks, and Stan Johnson NASA Johnson Space Flight Center/NASA Ames Research Center | 101 |
| Dimitri J. Mavriplis | <i>Solution of the Steady State Navier-Stokes Equations on Unstructured and Adaptive Meshes</i> NASA Langley Research Center/ICASE | 102 |
| Hans G. Mayr | <i>Numerical Study of Planetary Atmospheres</i> Co-investigator: Kwing L. Chan NASA Goddard Space Flight Center | 103 |
| Charles R. McClinton | <i>National Aero-Space Plane Propulsion Flow-Path Analysis and Code Certification</i> Co-investigators: R. Bittner, G. Bobskill, R. Hawkins, T. Jentink, P. Kamath, M. Mao, G. Mekkes, and D. Riggins NASA Langley Research Center | 104 |
| W. J. McCroskey | <i>Aerodynamic Flows about High-Performance Rotor Blade Tips</i> Co-investigators: J. D. Baeder, G. R. Srinivasan, and E. P. N. Duque U. S. Army Aeroflightdynamics Directorate—AVSCOM/NASA Ames Research Center | 105 |
| W. J. McCroskey | <i>Airloads and Acoustics of Rotorcraft</i> Co-investigators: J. D. Baeder, E. P. Duque, and G. R. Srinivasan U. S. Army Aeroflightdynamics Directorate—AVSCOM/NASA Ames Research Center | 106 |
| W. J. McCroskey | <i>Tilt-Rotor Aerodynamic Interactions</i> Co-investigators: V. Raghavan and S. Stanaway NASA Ames Research Center | 107 |
| S. Naomi McMillin | <i>Incipient Leading-Edge Separation</i> NASA Langley Research Center | 108 |
| N. Duane Melson | <i>Multiblock, Multigrid Method for the Solution of the Three-Dimensional Euler Equations</i> Co-investigators: Frank E. Cannizzaro, Alaa Elmiligui, and E. von Lavante NASA Langley Research Center | 109 |
| John E. Melton | <i>Computational Fluid Dynamics Analysis of Advanced Turboprop Configurations</i> Co-investigators: Ronald G. Langhi and Brian A. Nishida NASA Ames Research Center | 110 |
| Michael R. Mendenhall | <i>Vortex-Induced Nonlinearities on Submarines</i> Co-investigator: Stanley C. Perkins, Jr. Nielsen Engineering & Research, Inc. | 111 |
| Gene P. Menees | <i>Standing Oblique Detonation Waves</i> Co-investigators: Henry G. Adelman and Jean-Luc Cambier NASA Ames Research Center | 112 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|---|-------------|
| Suresh Menon | <i>Large-Eddy Simulations of Ramjet Combustion Instability</i> Co-investigator: Wen-Huei Jou QUEST Integrated, Inc. | 113 |
| Suresh Menon | <i>Mixing Enhancement for Scramjet Flameholders</i> Co-investigator: Emerick Fernando QUEST Integrated, Inc. | 114 |
| Charles L. Merkle | <i>Absorption of Microwave Energy in a Flowing Gas</i> Co-investigator: S. Venkateswaran The Pennsylvania State University | 115 |
| Christopher J. Miller | <i>Numerical Simulation of Advanced Propellers</i> Co-investigator: Saif A. Warsi NASA Lewis Research Center/Sverdrup Technology, Inc. | 116 |
| William H. Miller | <i>Quantum Mechanical Reactive Scattering</i> Co-investigator: John Z. H. Zhang University of California, Berkeley | 117 |
| Joseph H. Morrison | <i>Multi-Tasked Numerical Simulation of Complex Configurations in Hypersonic Flow</i> Co-investigator: David L. Whitaker NASA Langley Research Center | 118 |
| Robert D. Moser | <i>Wall-Bounded Turbulent Flows</i> Co-investigator: John Kim NASA Ames Research Center | 119 |
| Wolfgang Mueller | <i>Electronic Structure of Superconductors</i> Analatom, Inc./NASA Lewis Research Center | 120 |
| D. A. Naik | <i>Transonic Potential Flow about Transport Aircraft</i> Co-investigators: A. M. Ingraldi, R. G. Jorstad, T. A. Reyhner, and S. F. Yaros ViGYAN, Inc./NASA Langley Research Center/The Boeing Company | 121 |
| R. M. Nallasamy | <i>Unsteady Flow Field on an Advanced Propeller</i> Sverdrup Technology, Inc./NASA Lewis Research Center | 122 |
| David Nixon | <i>Pegasus™ Aerodynamic Analyses</i> Co-investigators: Gary D. Kuhn, Steven Caruso, and Michael R. Mendenhall Nielsen Engineering & Research, Inc. | 123 |
| Charles R. Olling | <i>Navier-Stokes Simulation of Separated Turbulent Flow around a Generic Fighter Aircraft</i> Co-investigators: Pradeep Raj and Josef S. Sikora Lockheed Aeronautical Systems Company | 124 |
| Charles R. Olling | <i>Navier-Stokes Simulation of Turbulent Flow over Rectangular Cavities Containing Stores</i> Co-investigators: Pradeep Raj and James E. Brennan Lockheed Aeronautical Systems Company | 125 |
| Elaine S. Oran | <i>Unsteady Diffusion Flames</i> Co-investigator: Janet L. Ellzey Naval Research Laboratory/Berkeley Research Associates | 126 |
| Ajay K. Pandey | <i>Flux-Based Finite-Element Formulation Reduces CPU Time for Thermal-Structural Analysis</i> Lockheed Engineering and Sciences Company/NASA Langley Research Center | 127 |
| V. C. Patel | <i>Hydrodynamics of Self-Propelled Bodies</i> Co-investigator: F. Stern Iowa Institute of Hydraulic Research/University of Iowa | 128 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|---|-------------|
| James L. Pittman | <i>Generic National Aero-Space Plane Afterbody</i> Co-investigators: Kenneth E. Tatum and Mohamed E. El-Eshaky NASA Langley Research Center | 129 |
| James L. Pittman | <i>NASP-TMP Forebody/Inlet Integration</i> Co-investigator: Lawrence D. Huebner NASA Langley Research Center | 130 |
| Richard H. Pletcher | <i>Development and Evaluation of Computational Methods for Turbomachinery Applications</i> Co-investigator: Edward J. Hall Iowa State University/Allison Gas Turbine Division, General Motors Corporation | 131 |
| Andrew Pohorille | <i>Vapor-Phase Growth of Crystals in Microgravity</i> University of California, Berkeley | 132 |
| Ramadas K. Prabhu | <i>Finite-Element Analysis of Equilibrium and Nonequilibrium Flows</i> NASA Langley Research Center/Lockheed Engineering and Sciences Company | 133 |
| Thomas H. Pulliam | <i>Chaotic, Unsteady, Low-Reynolds Number Navier-Stokes Flow</i> NASA Ames Research Center | 134 |
| Man Mohan Rai | <i>A Finite-Difference Approach to Direct Simulations and Large-Eddy Simulations of Turbulent Flow</i> NASA Ames Research Center | 135 |
| Man Mohan Rai | <i>Three-Dimensional Multi-Airfoil Navier-Stokes Simulations of Rotor/Stator Interaction in Turbines</i> Co-investigators: Nateri K. Madavan and Akil A. Rangwalla NASA Ames Research Center | 136 |
| Man Mohan Rai | <i>Unsteady Flow in a Multistage Compressor</i> Co-investigators: Karen L. Gundy-Burlet and Akil Rangwalla NASA Ames Research Center | 137 |
| M. S. Raju | <i>Analysis of Rotary Engine Processes and Computer Codes</i> Sverdrup Technology, Inc. | 138 |
| R. Ramakrishnan | <i>An Adaptation Procedure Combining Mesh Refinement with Mesh Movement for Compressible Flows</i> NASA Langley Research Center | 139 |
| David A. Randall | <i>Modeling Global Cloudiness</i> Colorado State University | 140 |
| James A. Rhodes | <i>Computation of ASTOVL Flow Fields</i> McDonnell Aircraft Company | 141 |
| Ben R. Riley | <i>Kinetic Theory Model for the Flow of a Simple Gas from a Three-Dimensional Axisymmetric Nozzle</i> University of Evansville | 142 |
| Charles L. Rino | <i>Scattering from Ocean Surfaces and Near-Surface Objects</i> Co-investigators: Hoc D. Ngo and Thomas L. Crystal Vista Research, Inc. | 143 |
| Michael M. Rogers | <i>Turbulent Reacting Flows</i> Co-investigators: Robert Moser and Chris Rutland NASA Ames Research Center | 144 |
| R. Clayton Rogers | <i>Hydrogen-Air Combustion Simulation in a Pulse Facility</i> Co-investigator: Elizabeth H. Weidner NASA Langley Research Center | 145 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|---|-------------|
| William C. Rose | <i>National Aero-Space Plane Inlet Boundary Layer Control, Phase II</i> NASA Lewis Research Center/Rose Engineering & Research, Inc. | 146 |
| William C. Rose | <i>National Aero-Space Plane Inlet Flow Fields</i> Co-investigator: Edward W. Perkins NASA Ames Research Center/Rose Engineering & Research, Inc. | 147 |
| William C. Rose | <i>Three-Dimensional Viscous Flow in High-Speed Inlets</i> Co-investigator: Edward W. Perkins NASA Ames Research Center/Rose Engineering & Research, Inc. | 148 |
| M. B. Salamon | <i>Advanced Computational Materials</i> Co-investigators: R. Averbach, Y.-C. Chang, J. Kogut, R. Martin, and P. Wolynes University of Illinois, Urbana-Champaign | 149 |
| Matthew T. Scott | <i>Computational Methods for Rotor-Blade Drag Prediction</i> Bell Helicopter Textron, Inc. | 150 |
| Balu Sekar | <i>Direct Simulation of High-Speed Mixing Layers With and Without Chemical Heat Release</i> Co-investigators: H. S. Mukunda and Mark H. Carpenter NASA Langley Research Center | 151 |
| John V. Shebalin | <i>Ionized Flow Around a Reentry Vehicle</i> NASA Langley Research Center | 152 |
| Peter K. Shih | <i>Afterbody Aerothermodynamic Studies for the Hypersonic Glide Vehicle</i> Co-investigator: Che-Shing Kang General Dynamics, Convair Division | 153 |
| Jian-Shun Shuen | <i>Advanced Numerical Algorithms for Chemically Reacting Flows</i> Sverdrup Technology, Inc./NASA Lewis Research Center | 154 |
| William A. Sirignano | <i>Ignition and Flame Spread above Liquid Fuel Pools: Gravity Effects</i> Co-investigators: Fanghei Tsau and David N. Schiller University of California, Irvine | 155 |
| Philippe R. Spalart | <i>Transition and Turbulence in Three-Dimensional Flows</i> NASA Ames Research Center | 156 |
| Frank J. Spera | <i>Magma Chamber Convection</i> University of California, Santa Barbara | 157 |
| B. M. Steinetz | <i>National Aero-Space Plane Engine Corner-Flow</i> Co-investigators: W. J. Coirier and Mike Tong NASA Lewis Research Center/Sverdrup Technology, Inc. | 158 |
| Roger C. Strawn | <i>Computational Fluid Dynamics Prediction of Advanced Rotor Performance</i> Co-investigators: John Bridgeman, K. Ramachandran, and Frank Caradonna U.S. Army Aeroflightdynamics Directorate—AVSCOM/NASA Ames Research Center | 159 |
| Craig L. Streett | <i>Numerical Simulation of Wave Interactions Related to Transition</i> Co-investigator: S. Balachandar NASA Langley Research Center | 160 |
| Craig L. Streett | <i>Transition Simulations in Nonhomogeneous Geometries</i> NASA Langley Research Center | 161 |
| Phil C. Stuart | <i>Mars Rover/Sample Return Aerocapture Vehicle</i> Co-investigator: Chien-Peng Li NASA Johnson Space Center | 162 |

| Principal Investigator | NAS Summary | Page |
|-------------------------------|--|-------------|
| Sundaresa V. Subramanian | <i>Numerical Study of Three-Dimensional Viscous Reacting Flows with Application to Scramjet Propulsion</i> Co-investigator: Michael J. Epstein General Electric Aircraft Engines | 163 |
| Dennis Sullivan | <i>Microwave Hyperthermia Computer Modeling</i> Co-investigator: Bryan James Stanford University School of Medicine | 164 |
| Chao-Ho Sung | <i>Naval Applications of Computational Fluid Dynamics</i> Co-investigator: Michael J. Griffin David Taylor Research Center | 165 |
| R. C. Swanson | <i>Development of Algorithms for Solving the Three-Dimensional Navier-Stokes Equations</i> Co-investigator: M. Sanetrik NASA Langley Research Center | 166 |
| Tsze C. Tai | <i>Hydrodynamic Prediction Methods for Advanced Submarine Configurations</i> Co-investigators: Steven C. Fisher, Cheng-Wen Lin, and Gerald D. Smith David Taylor Research Center | 167 |
| Tsze C. Tai | <i>Low-Speed Maneuver Aerodynamics in a Nonuniform Free Stream</i> David Taylor Research Center | 168 |
| Ronald K. Takahashi | <i>Unsteady Euler Analysis of the Redistribution of an Inlet Temperature Distortion in a Turbine</i> Co-investigator: Ron-Ho Ni United Technologies, Pratt & Whitney | 169 |
| Luen T. Tam | <i>Thermochemical and Radiative Nonequilibrium Flow Simulation for the Aeroassist Flight Experiment</i> Co-investigator: Chien P. Li Lockheed Engineering and Sciences Company/NASA Johnson Space Center | 170 |
| John C. Tannehill | <i>Development of a Robust Parabolized Navier-Stokes Code for Computing Three-Dimensional, Chemically Reacting Flow Fields</i> Co-investigators: Phil E. Buelow and John O. Ivaldi Iowa State University | 171 |
| Rajiv Thareja | <i>Computation of Three-Dimensional Flows Using Unstructured Grids</i> Co-investigators: Ken Morgan, Jaime Peraire, Obay Hassan, and Joaquin Peiro NASA Langley Research Center | 172 |
| Rajiv Thareja | <i>Viscous Hypersonic Flow over a 24-Degree Compression Corner</i> Co-investigators: Ken Morgan and Jaime Peraire NASA Langley Research Center | 173 |
| Donald G. Truhlar | <i>Scattering Theory and Calculations for Chemical Reactions and Molecular Energy Transfer</i> Co-investigators: David C. Chatfield, Philippe Halvick, David W. Schwenke, Thanh N. Truong, Michael J. Unekis, and Meishan Zhao University of Minnesota | 174 |
| Pratap Vanka | <i>Hot Gas Ingestion by STOVL Aircraft</i> Co-investigators: D. K. Tafti and W. Pegeus University of Illinois, Urbana-Champaign | 175 |
| Bram vanLeer | <i>Calculation of Hypersonic Flows with Strong Surface Blowing</i> Co-investigators: T. C. Adamson, A. F. Messiter, N. D. Carter, and M. D. Matarrese The University of Michigan | 176 |
| Veer N. Vatsa | <i>High Reynolds Number, Transonic, Viscous Flow over Aircraft Components</i> Co-investigator: Bruce W. Wedan NASA Langley Research Center | 177 |

| Principal Investigator | NAS Summary | Page |
|--------------------------------------|---|-------------|
| Bill J. Walker | <i>Tactical Missile Aero-Propulsion Interaction</i> Co-investigators: C. D. Mikkelsen, K. D. Kennedy, K. L. Cornelius, and M. E. Vaughn, Jr. U.S. Army Missile Command—Redstone Arsenal | 178 |
| Robert W. Walters | <i>Development of Unstructured and Nonequilibrium Chemistry Upwind Algorithms for Hypersonic Flows</i> Co-investigators: Bernard Grossman, P. Cinnella, Andrew S. Godfrey, William D. McGrory, and David C. Slack Virginia Polytechnic Institute and State University | 179 |
| Jong H. Wang | <i>Navier-Stokes Predictions of Hypersonic Chemically Reacting Flows</i> Co-Investigators: David Yeh and Mike George Rockwell International, North American Aircraft Division | 180 |
| Kurt F. Weber | <i>Simulation of Three-Dimensional Flow in a Transonic Fan Rotor</i> Co-investigator: Dale W. Thoe Allison Gas Turbine Division, General Motors Corporation | 181 |
| Kenneth J. Weilmuenster | <i>Winged Entry Vehicle Computations</i> Co-investigator: Francis A. Greene NASA Langley Research Center | 182 |
| Robert A. West | <i>Optical Properties of Nonspherical Particles in Planetary Atmospheres</i> Jet Propulsion Laboratory/California Institute of Technology | 183 |
| Thomas C. Wey | <i>Application of Unstructured Grids to Shuttle Computational Fluid Dynamics</i> Co-investigator: Chien-Peng Li Lockheed Engineering and Sciences Company/NASA Johnson Space Center | 184 |
| Richard G. Wilmoth | <i>Shock Interactions in Rarefied Hypersonic Flows</i> Co-investigator: Bradford Sturtevant NASA Langley Research Center/California Institute of Technology | 185 |
| Chung-Jin Woan | <i>Computational Fluid Dynamics Design of a Laminar-Flow Control Glove</i> Co-investigator: Michael W. George Rockwell International, North American Aircraft Division | 186 |
| Cheng-I Yang | <i>Numerical Simulation of Submarine Propulsion</i> David Taylor Research Center | 187 |
| Ruey-Jen Yang | <i>Multistage Rotor/Stator Interaction in the Space Shuttle Main Engine Turbopump Turbine</i> Co-investigator: Shyi-Jang Lin Rockwell International, Rocketdyne Division | 188 |
| Larry Young | <i>Tilt-Rotor Download Prediction</i> Co-investigator: C. S. Lee NASA Ames Research Center | 189 |
| Richard E. Young | <i>Simulation of the Climatology of the El Chichon Volcanic Aerosol Cloud in the Stratosphere</i> Co-investigators: Owen B. Toon and James B. Pollack NASA Ames Research Center | 190 |
| N. J. Yu | <i>Development of an Euler-Based Method for Turboprop Integration</i> Co-investigators: H. C. Chen, T. J. Kao, and D. A. Naik The Boeing Company/VIGYAN, Inc./NASA Langley Research Center | 191 |
| T. A. Zang | <i>Incompressible Transition to Turbulence</i> Co-investigators: S. Dinavahi, B. Singer, and U. Piomelli NASA Langley Research Center | 192 |
| Index by Research Sites | | 193 |

NAS Technical Summaries

Large-Scale Ocean Modeling

David Adamec, Principal Investigator

Co-investigators: Michele M. Rienecker and Antonio J. Busalacchi

NASA Goddard Space Flight Center

Research Objective

To develop tools that allow simulations of ocean flows, which help in understanding the role of the oceans in the Earth climate system.

Approach

Three-dimensional quasi-geostrophic and primitive-equation ocean models were used to examine ocean mesoscale variability in the midlatitudes. Multigrid techniques were developed to help speed model calculations.

Accomplishment Description

Three-dimensional, linear stability analyses were used to study the stability of various Gulf Stream flow paths. In particular, the following aspects of an assumed basic flow were studied, as they relate to the sensitivity of growing instabilities: meander wavelength, meander steepness, trough width and steepness, ridge width and steepness, strength of initial flow, and ratio of barotropic to baroclinic energy in the basic flow. The accompanying figure shows the results of a stability analysis for a flow path with a narrow trough. The initial flow is indicated by the white arrows. Both the three-dimensional surface and the color scheme indicate the magnitude of the fastest growing instability. The instability with the largest magnitude occurs

just downstream of the trough crest. Each analysis requires 45 Cray Y-MP minutes and 10 megawords of memory. Also, a new three-dimensional computer model based on the differentiated form of the momentum equations was coded, tested, and optimized for Cray Y-MP use. This new code also uses our own newly developed multigrid solvers for spherical coordinates.

Significance

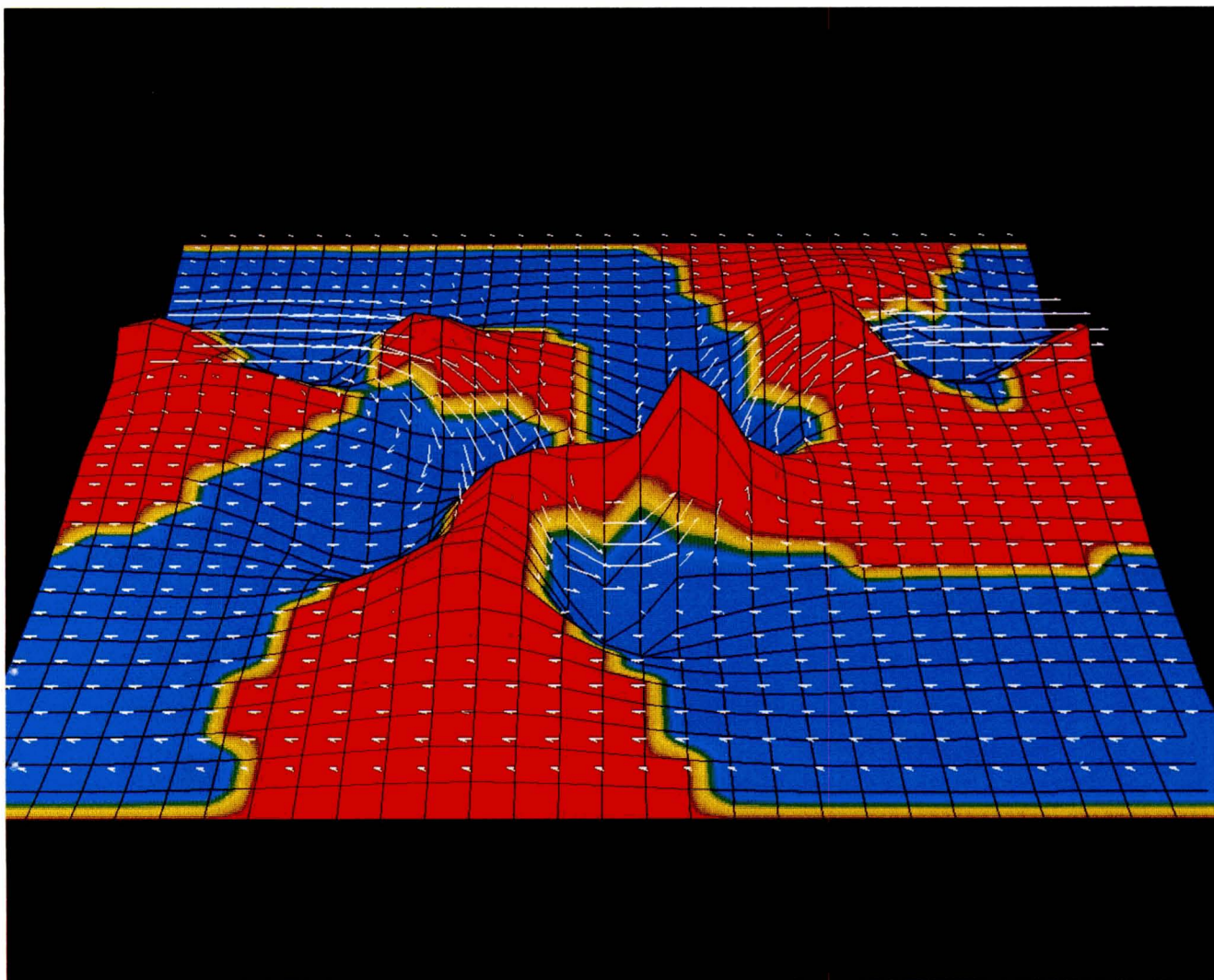
The exact role of the ocean in the meridional exchange of heat and momentum is not well understood. These types of ocean simulations and calculations help in understanding and quantifying the effect of the ocean on the Earth climate system.

Future Plans

The three-dimensional stability analyses will be continued using Gulf Stream paths determined from satellite data, and the new primitive-equation code will be used to study the formation and maintenance mechanisms of ocean fronts.

Publications

"Three-Dimensional Stability of Gulf Stream Flow." Presented at the Goddard Space Flight Center Ocean Seminar Series, Feb. 1990.



Stability analysis for a flow path with a narrow trough.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Helicopter Fuselage and Rotor Flow-Field Calculations in Hover and Forward Flight

Ramesh K. Agarwal, Principal Investigator

Co-investigator: Jerry E. Deese

McDonnell Douglas Research Laboratories/NASA Ames Research Center

Research Objective

To develop computational codes capable of predicting the transonic viscous flow about helicopter fuselages and multi-bladed helicopter rotors both in hover and in forward flight.

Approach

For calculating turbulent flow about helicopter fuselages, the Reynolds-averaged Navier-Stokes equations are solved in an inertial frame on a body-conforming curvilinear grid using a finite-volume explicit Runge-Kutta time-stepping scheme. For calculating turbulent flow about rotor blades, Reynolds-averaged Navier-Stokes equations are solved in a rotating coordinate system. Rotor-wake effects are modeled in the form of a correction applied to the geometric angle of attack along the blades. This correction is obtained by computing the local induced downwash with a prescribed- or free-wake analysis program.

Accomplishment Description

For helicopter fuselage flow-field calculations, the Reynolds-averaged Navier-Stokes code MDNS3D was installed on a Cray-2. Flow fields about an Apache and an MDX fuselage were computed. The accompanying figure shows the pressure distribution on the MDX fuselage. A typical calculation on a $112 \times 20 \times 20$ grid required 6 megawords (64 bits each) of Cray-2 memory and approximately 2.5 hours of CPU time. For rotor flow-field calculations, the Euler/thin-layer rotor code MDROTH was used. Calculations were performed for the flow field about the Gazelle rotor and about a rotor with a British Experimental Rotor Program (BERP) -like tip in forward flight (Publications 1 and 2). A typical calculation on a $97 \times 33 \times 21$

grid required 8 megawords (64 bits each) of Cray-2 memory, and approximately 40 minutes of CPU time for hover and 8 hours of CPU time for forward flight.

Significance

The development of MDNS3D and MDROTH codes would be of great value in the design and development of rotorcraft. A computational data base consisting of a number of blade shapes at various flight conditions could significantly reduce the time and cost for configuration selection.

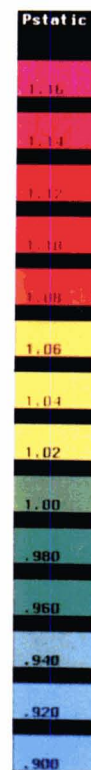
Future Plans

The main focus will be on the calculation of rotor/body interaction flow fields. MDNS3D and MDROTH will be combined for this purpose. Fine-grid calculations will be performed on multi-bladed helicopter rotor configurations with unconventional blade tips, including the BERP and Puma tips, and the results will be compared with the experimental data whenever possible. Calculations will also be performed for a no-tail-rotor (NOTAR) configuration and a trail-rotor-convertible (TRC) configuration.

Publications

1. Agarwal, R. K., and Deese, J. E. "Euler/Navier-Stokes Calculations of the Flowfield of a Helicopter Rotor in Hover and Forward Flight." *Proceedings of the Second International Conference on Rotor Craft Basic Research*. Univ. of Maryland, Feb. 1988.
2. Agarwal, R. K., and Deese, J. E. "Navier-Stokes Calculations of the Flowfield of a Helicopter Rotor in Hover." AIAA Paper 88-0106, Jan. 1988.

HELICOPTER FUSELAGE PREDICTIONS
 $M_{inf} = 0.4$ AOA = -5.0° RE = 60 MILLION



Pressure distribution on the MDX fuselage; $M_{inf} = 0.4$, $\alpha = -5.0^\circ$, $Re = 60 \times 10^6$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Numerical Solution of the Reynolds-Averaged Navier-Stokes Equations for Flow About an Almost Complete Aircraft

Ramesh K. Agarwal, Principal Investigator

Co-investigator: Jerry E. Deese

McDonnell Douglas Research Laboratories/NASA Ames Research Center

Research Objective

To develop a computational code capable of predicting the viscous flow field over a complete aircraft.

Approach

Reynolds-averaged Navier-Stokes equations are solved on a global body-conforming curvilinear grid by using a finite-volume Runge-Kutta time-stepping scheme. The global grid about the aircraft is generated by using a three-dimensional grid-generation code based on the blended grid-generation technique. Turbulence effects are modeled using a simple algebraic eddy-viscosity model. Variable time steps and implicit smoothing of the residuals are used to enhance convergence to steady state.

Accomplishment Description

The Euler/Reynolds-averaged Navier-Stokes code MDAIRPLANE was installed on a Cray-2. Of a number of significant accomplishments, especially noteworthy are transonic flow-field calculations about complete F-15 and MD-80 configurations and T-45, MD-11, and MD-91 wing/body configurations (Publications 1 and 2). Computations compared well with experimental data. The accompanying figure shows the pressure distribution on an MD-80 aircraft. A typical calculation was performed on a $160 \times 34 \times 42$ mesh and required 26 megawords (64 bits each) of Cray-2 memory and approximately 10 hours of CPU time.

Significance

The development of the MDAIRPLANE code would be of great value in the design and development of transport and fighter aircraft, because a greater number of candidate vehicle configurations could be considered and their performance over a wide range of flight conditions could be evaluated, thereby reducing the cost and time required in the design cycle.

Future Plans

Three-dimensional grids will be generated for a complete MD-11 transport, a C-17 transport, and an F-18 fighter aircraft. Calculations will be performed using fine grids and requiring approximately 32 megawords (64 bits each) of Cray-2 memory. The results will be compared with experimental data for transonic Mach numbers.

Publications

1. Agarwal, R. K.; Deese, J. E.; Johnson, J. G.; and Steinhoff, J. S. "Euler Calculations for Flow Over a Complete Aircraft." AIAA Paper 89-2221, 1989.
2. Deese, J. E., and Agarwal, R. K. "Turbulent Flow Calculations for Flow Over Wings Near Maximum Lift." AIAA Paper 89-2239, 1989.

TRANSPORT AIRCRAFT PREDICTIONS

$$M_{inf} = 0.76$$

$$AOA = 2.0$$



Pressure distribution on an MD-80 aircraft; $M_{inf} = 0.76$, $\alpha = 2.0^\circ$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Separated and Vortical Flow-Field Analysis on Fighter Wing Configurations

Shreekant Agrawal, Principal Investigator

Co-investigators: Tom A. Kinard and Brian A. Robinson

McDonnell Aircraft Company

Research Objective

One objective of this study is to perform an evaluation of various Navier-Stokes solvers using different solution algorithms for transonic wing flow-field calculations. The evaluation focuses on accuracy, efficiency, speed, and convergence, and seeks areas for future research. A second objective (not described here) is to analyze differences between Euler and Navier-Stokes solutions on a flat-plate delta wing at high angles of attack.

Approach

Three different Navier-Stokes solvers are used to predict transonic flow fields on the ONERA M6 wing for two flow conditions, attached flow ($M = 0.84$, angle of attack = 3.06°) and separated flow ($M = 0.837$, angle of attack = 6.06°). The three flow solvers are CFL3D, TLNS3D, and TNS.

Accomplishment Description

A C-O-type grid of dimensions 193 (streamwise), 49 (normal), and 33 (spanwise) was used for the CFL3D and TLNS3D codes. Because of the four-zone structure of the TNS code, the dimensions and viscous stretching parameters had to be adjusted to yield a similar number of surface grid points and points in the boundary layer as the C-O grid. The turbulent eddy viscosity for all solutions was computed using the Baldwin-Lomax model. The TLNS3D solutions were also obtained with a Johnson-King turbulence model. Computed surface pressure coefficients are compared with experimental data in the accompanying figure. For attached flow, similar solutions are found with the different codes, except for some differences near shock waves and leading-edge regions; the scatter in the shock wave locations is approximately 6%. For

separated flow, however, the scatter band of the solutions from the different codes is considerably larger; shock strengths and locations are predicted very differently. With the Johnson-King turbulence model, a remarkable improvement in the TLNS3D solution is noticeable. The upper-surface particle traces due to the three codes are also shown. The CPU time for TLNS3D was 1.5 Cray-2 hours for the attached-flow case and 2.0 hours for the separated-flow case. The CPU times for TNS and CFL3D were about two times longer than for TLNS3D. The amount of central memory used for an average * job run was 20 megawords.

Significance

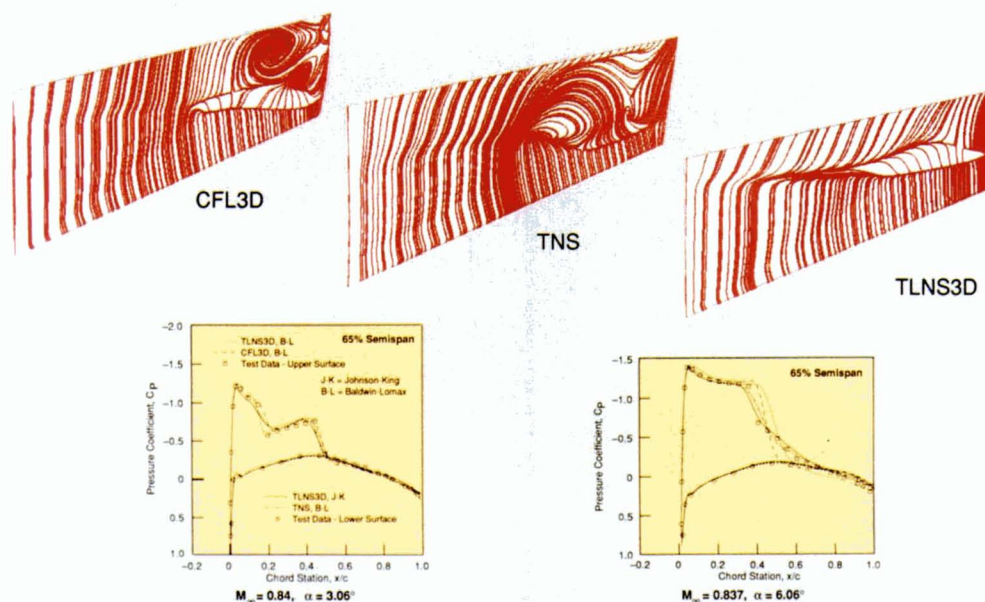
Various three-dimensional Navier-Stokes codes differ from each other in several respects, such as numerical algorithms, boundary condition treatment, dissipation models, convergence acceleration techniques, and turbulence models. Because of these differences, the results obtained from various codes differ from each other, even though they solve the same equations of fluid mechanics. Therefore, in order to select the most accurate and efficient flow solver for a particular application, an evaluation of the codes is important.

Future Plans

The performance of the codes will be further evaluated on more complete (e.g., wing-fuselage) aircraft geometries.

Publications

Agrawal, Shreekant; Barnett, Matt; and Robinson, Brian A. "Investigation of Vortex Breakdown on a Delta Wing Using Euler and Navier-Stokes Equations." Presented at the AGARD Symposium on Vortex Flow Aerodynamics, Scheveningen, Netherlands, Oct. 1990.



GP03-0007-221

(Top) Particle traces on upper surface of the ONERA M6 wing; $M_\infty = 0.8337$, $\alpha = 6.06^\circ$, $Re = 11.7 \times 10^6$ based on MAC, Baldwin-Lomax turbulence model. The CFL3D and TLNS3D grids are $193 \times 49 \times 33$. The TNS four-zone grid is Zone 1, $101 \times 33 \times 25$; Zone 2, $101 \times 37 \times 21$; Zone 4, $93 \times 35 \times 25$. (Bottom) Surface pressure coefficients.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Discrete Particle Simulation of Compressible Flow

Donald Baganoff, Principal Investigator

Co-investigators: William J. Feiereisen, Jeffrey McDonald, Brian Haas, Michael Woronowicz, Michael Fallavollita, and Leo Dagum
Stanford University/NASA Ames Research Center

Research Objective

To develop, test, and extend a new particle simulation method for rarefied hypersonic flow, and to assess its ability to treat realistic three-dimensional geometries.

Approach

Direct particle simulation methods model a rarefied flow as a large collection of discrete particles that interact with each other through collisions. The well-known collision-selection rule used in the direct simulation Monte Carlo (DSMC) method is replaced by a new statistical rule that is highly compatible with the requirements for vectorization. Many algorithmic improvements that go beyond those that are related only to vectorization issues are also used.

Accomplishment Description

The collision-selection rule was shown to give results that were identical to those of the DSMC method in predicting shock-wave structure and in predicting the correct mean-free-path variation with density and temperature, for power-law interactions ranging from hard sphere to Maxwell molecule. A single-species version of the vectorized code, which assumes an ideal diatomic gas, was used to study flow in a flat-plate boundary layer, flow past a wedge, and full three-dimensional flow past the Aeroassist Flight Experiment (AFE) aircraft body. The accompanying figure depicts the pressure distribution in the central plane of the body and in a downstream cross-section of the wake, giving an approximate image of the three-dimensional structure of the flow. The simulated conditions correspond to a Mach number of 35 at an altitude of 100 km, assuming a 4.25-m-diameter body. The simulation used 9.5 million particles and 0.43 million cells. The run required 4.5 hours of Cray-2 CPU time and 125 megawords of central

memory. The performance of the code is presently 1 to 2 microseconds per particle per time step.

Significance

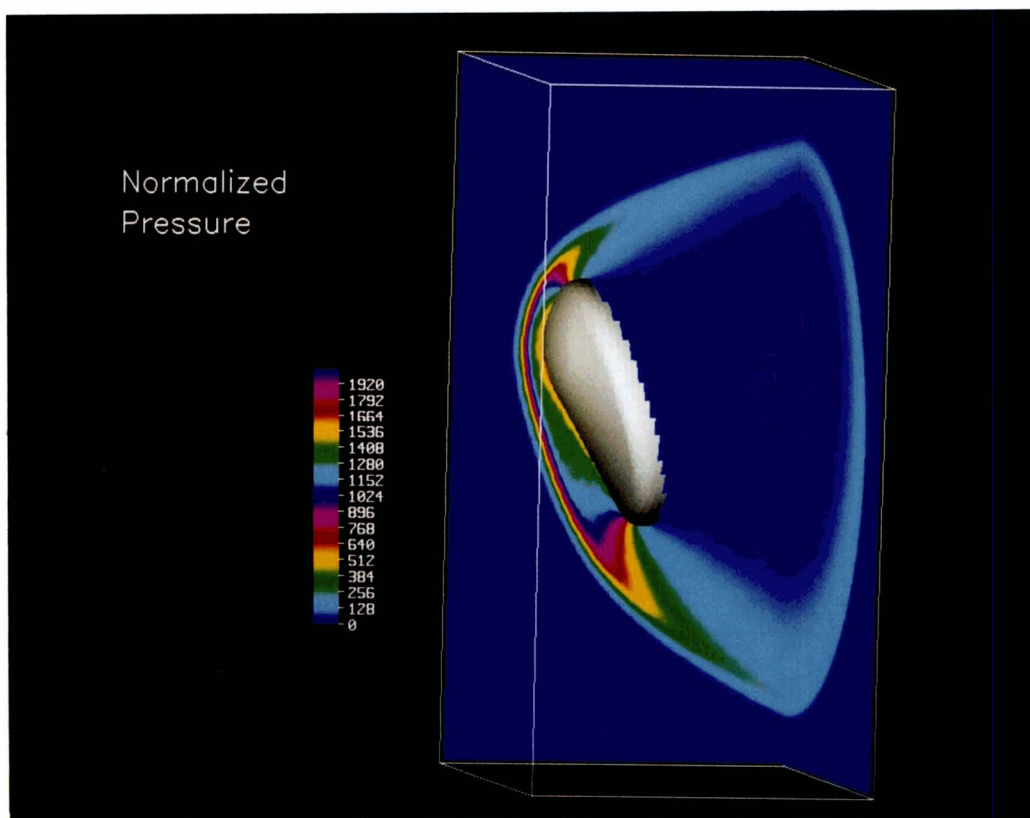
With the increasing interest in hypersonic flight, a need arises to develop a computational predictive capability for the aerodynamic and thermal environment found around vehicles such as the National Aero-Space Plane or the AFE. Because of the difficulties in applying the continuum equations to the very low density hypersonic regimes these vehicles will encounter, it is appropriate to consider the application of direct particle simulation methods. The numerical efficiency of the current approach enables direct particle simulations to be carried out on a much larger scale than previously possible.

Future Plans

A new version of the vectorized code, which includes provisions for multiple species and rotational and vibrational nonequilibrium, as well as chemical reactions, is being prepared, and early indications are that its performance will be comparable with the present single-species code.

Publications

1. Feiereisen, W. J., and McDonald, J. D. "Three-Dimensional Discrete Particle Simulation of an AOTV." AIAA Paper 89-1711, 1989.
2. Woronowicz, M. S., and McDonald, J. D. "Application of a Vectorized Particle Simulation in High-Speed Near-Continuum Flow." AIAA Paper 89-1665, 1989.
3. McDonald, J. D. "A Computationally Efficient Particle Simulation Method Suited to Vector Computer Architectures." SUDAAR No. 589, Stanford Univ., 1989.



Pressure distribution in the central plane of the body, and downstream.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Direct Simulation of Large-Eddy Breakup Devices

R. Balasubramanian, Principal Investigator
Spectrex Inc.

Research Objective

The ultimate objective of this work is to obtain a data base of numerically generated turbulence data for complex geometries such as large-eddy breakup (LEBU) devices, using a nonperiodic, fully spectral, Navier-Stokes complex-geometry multidomain algorithm.

Approach

A patched spectral element technique forms the basis for the direct-simulation solver; it uses spectral algorithms (Chebyshev polynomials in two directions and Fourier spectral representations in the third direction) with direct enforcement (Green's function method) of inter-element continuities at patch boundaries. The patched-element approach makes possible the application of a flow code to the flow over complex geometries, such as LEBU geometries, by proper decomposition of the computational domain to appropriate spectral patches.

Accomplishment Description

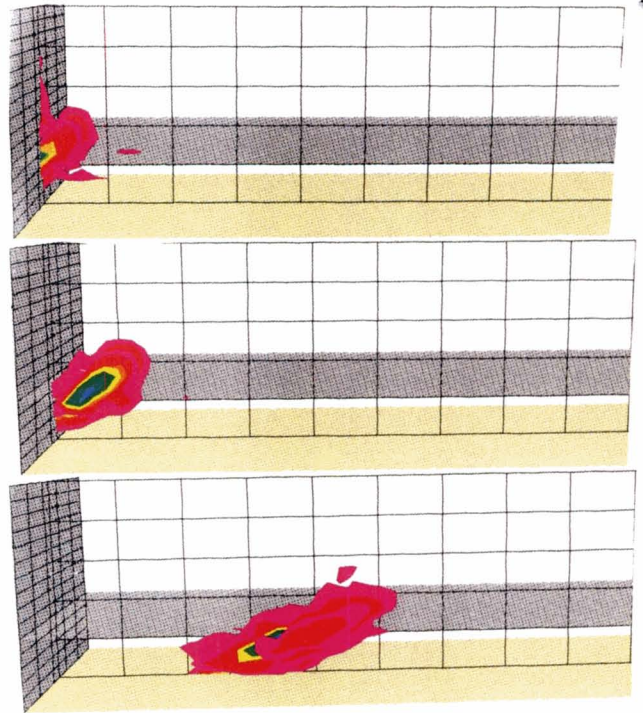
During this phase of the work, the patched-spectral-element algorithm was used to gain understanding of the flow physics of vortex interactions in shear flows. This engendered a clearer understanding of how the large-scale structures that populate turbulent outer layers sustain themselves by interaction with the wall-layer structures. The role of a predominantly longitudinal, transverse, or normal vortical structure interacting with shear flow was carefully documented by numerical flow visualization. Also, appropriate initial data bases, originating from Spalart's direct-simulation data base, have been developed and used for direct simulation of turbulence in the complex geometry case. These studies are still in their infancy.

Significance

Direct simulation of turbulence using nonperiodic boundary conditions is a relative newcomer to the field of computational fluid mechanics; patching techniques that we have developed allow the examination of distances further downstream than previously thought possible. Patching also makes possible the study of highly complex flow fields by direct simulation through appropriate geometric discretizations of the flow modules. Thus, direct comparisons of laboratory flows with computer-generated turbulence data may one day be possible using these high-order flow modules.

Future Plans

The patched-spectral-element code will be further used to extend Spalart's turbulence data from $Re = 1300$ to higher Reynolds numbers, using patches in the downstream direction. Complex-flow turbulence data for LEBU devices at intermediate Reynolds numbers will also be developed. Plans are also under way to incorporate compressibility effects, so that studies can be made of the suppression of turbulent noise by LEBU devices with their attendant ASW implications.



Interaction of an elliptical ring vortex in shear flow at various times of evolution. Contours of total vorticity are color coded. The wall, the $H = \delta$ planes, and the inflow plane are shaded.

Validation of a Three-Dimensional Flux-Split Euler Algorithm for Unstructured Grids

John T. Batina, Principal Investigator
Co-investigators: Elizabeth M. Lee, Russ D. Rausch, and William L. Kleb
NASA Langley Research Center

Research Objective

New algorithms are being developed for the solution of the three-dimensional unsteady Euler equations, based on the use of unstructured grids. These algorithms require validation before they can be used with confidence for general applications, so the objective of this research was to determine the accuracy of a new flux-split method of solving the Euler equations on an unstructured grid of tetrahedra.

Approach

The new algorithm contains recently developed improvements to the spatial and temporal discretizations used by unstructured-grid flow solvers. The spatial discretization now involves a flux-split approach, which is naturally dissipative and captures shock waves sharply with only one grid point within the shock structure. The temporal discretization uses either explicit or implicit time marching. The explicit time marching is a four-stage Runge-Kutta integration and the implicit time marching is a two-sweep Gauss-Seidel relaxation procedure. The algorithm is validated by performing calculations using the explicit integration for a well-defined wing (an AGARD standard configuration) and making comparisons with experimental steady-pressure data.

Accomplishment Description

To assess the accuracy of the new three-dimensional flux-split Euler solver, calculations were performed for the ONERA M6 wing. The M6 wing has a leading-edge sweep angle of 30° , an aspect ratio of 3.8, and a taper ratio of 0.562. The airfoil section of the wing is the ONERA "D" airfoil, which is a conven-

tional section with a 10% maximum thickness-to-chord ratio. The results were obtained using a grid that has 154,134 nodes and 869,056 tetrahedra. The surface triangulation for the upper surface of the wing is shown in the left half of the accompanying figure. Results were obtained for the M6 wing at a free-stream Mach number of 0.84 and an angle of attack of 3.06° . These conditions were chosen for comparison with experimental pressure data, as shown in the right half of the figure. Because of the large number of tetrahedra used in this case, the solution required approximately 150 Cray-2 CPU hours and 125 megawords of memory.

Significance

The Euler results are in good agreement with the experimental pressure data, especially in predicting the strength and location of the shock waves; this tends to verify the new Euler algorithm. The shocks are sharply captured with only one grid point within the shock structure, and there are no overshoots or undershoots.

Future Plans

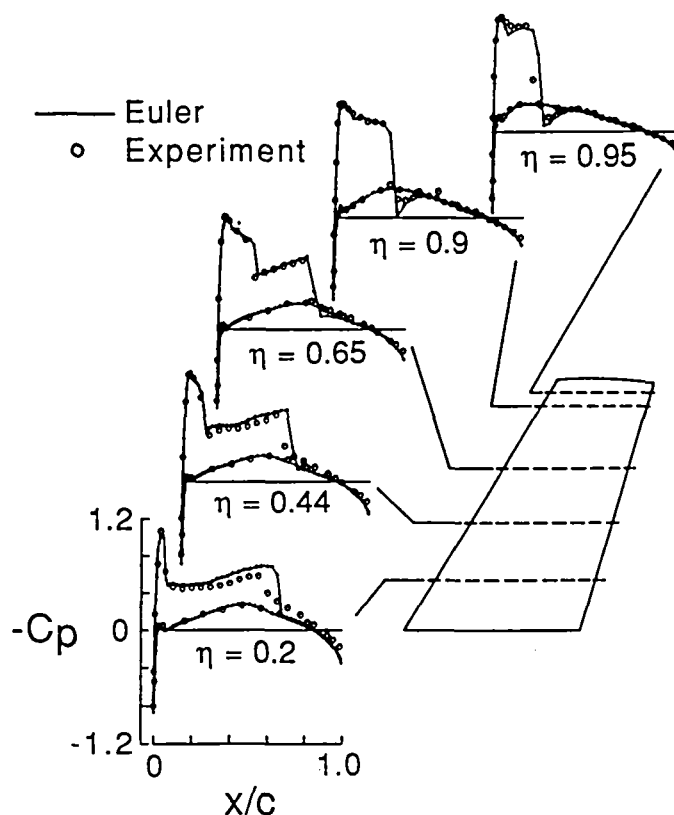
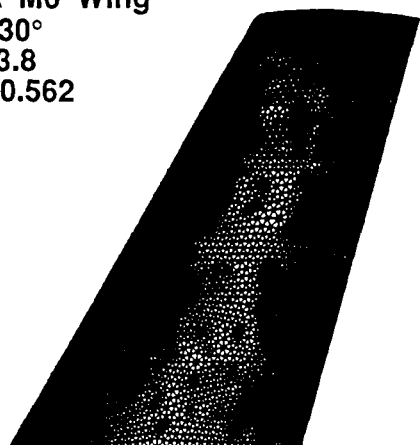
Unsteady applications will be performed to determine the accuracy as well as the efficiency of the new flux-split algorithm for such cases.

Publications

Batina, John T. "Accuracy of an Unstructured-Grid Upwind-Euler Algorithm for the ONERA M6 Wing." Presented at the Accuracy of Unstructured Grid Techniques Workshop, NASA Langley Research Center, Jan. 1990.

ONERA M6 Wing

$\Lambda = 30^\circ$
AR = 3.8
TR = 0.562



(Left) Surface triangulation for the upper surface of the ONERA M6 wing. (Right) Comparison of computed pressures with experimental pressures; $M = 0.84$, $\alpha = 3.06^\circ$.

Calculations of Unsteady, Supersonic-Jet Flow Fields and Acoustics

William W. Bower, Principal Investigator
Co-investigators: Robert E. Childs and Gerald E. Chmielewski
McDonnell Douglas Research Laboratories/Nielsen Engineering & Research, Inc.

Research Objective

To predict the dynamic behavior and acoustic fields associated with supersonic free and impinging jets.

Approach

The generation and radiation of aerodynamic noise is governed by the unsteady, compressible Navier-Stokes equations. Therefore, numerical solutions of the Navier-Stokes equations should yield predictions of the acoustic field in addition to the flow field, provided that the dominant physical phenomena of the flow that affect noise are resolved in the simulation. The present work is an effort to predict the flow field and noise generation for supersonic jets by means of numerical simulations based on the Navier-Stokes equations.

Accomplishment Description

Numerical simulations were performed for two flows: a free jet with a nozzle pressure ratio (NPR) of unity and a jet Mach number (M) of 1.19; and an impinging jet with NPR = 1.0, M = 1.46, and a height above ground of four nozzle diameters. The Reynolds number based on nozzle diameter for both cases was 70,000, and the jets were cold, having approximately the same stagnation temperature as the ambient fluid. The accompanying figure illustrates the pressure and vorticity in the free jet at a given time in the simulation. Small, randomly spaced vortices form not far downstream of the nozzle exit and merge, commonly in pairs, until the vortices are comparable in size to the jet radius. At this stage the axisymmetric

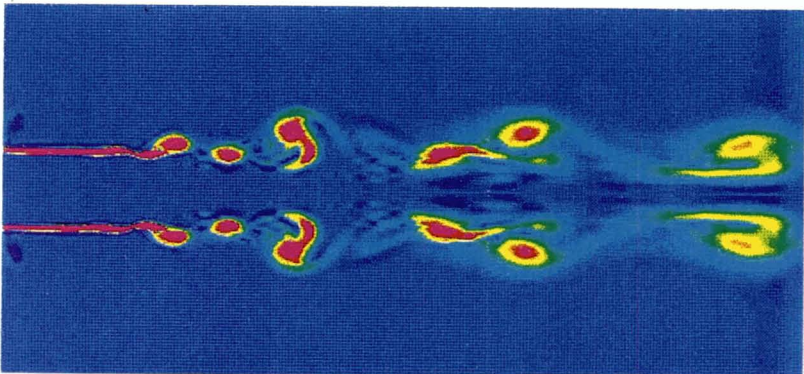
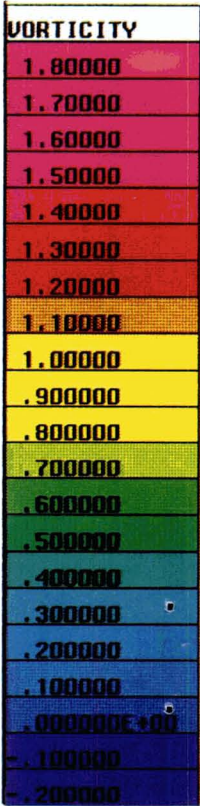
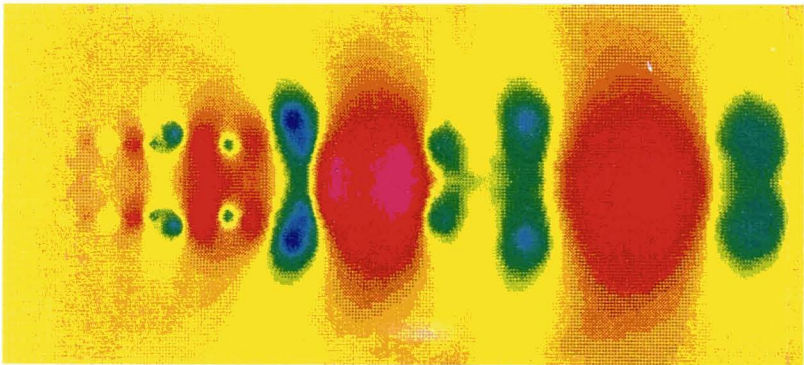
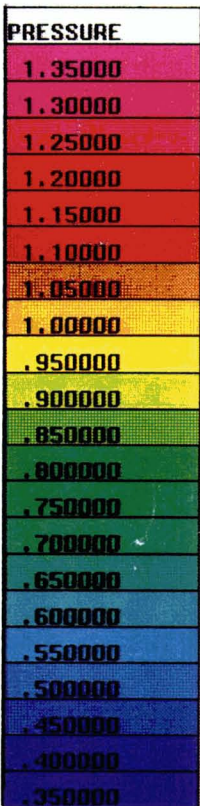
structure is saturated and can grow no further. In the impinging jet case, the higher Mach number reduces the growth rate of the axisymmetric mode. However, the impingement process produces a strong acoustic signal that forces the initial shear layer at frequencies below its natural instability frequency. The simulation shows that a large vortex is formed about one nozzle diameter downstream of the nozzle exit, and there is no vortex pairing before the structure encounters the impingement plane. Two noise-generation mechanisms associated with the impingement process are observed. When a vortex strikes the surface, its major axis expands; this stretches the ring vortices, intensifying the vorticity and significantly reducing the pressure at the vortex core. Also, the impingement shock is unsteady. Each simulation required approximately 1 Cray Y-MP hour and 1 megaword of central memory.

Significance

The present simulation provides insight into the sources of noise for supersonic jets and therefore can provide guidance for the development of techniques to reduce the jet acoustic signatures.

Future Plans

Continuation of this work will include comparisons of predicted overall sound-pressure levels with experimental data for supersonic jets, and an extension of the simulations to three dimensions.



Color contours of instantaneous pressure (top) and vorticity (bottom) for the unsteady, axisymmetric free jet in quiescent air; M = 1.19, Re = 70,000, NPR = 1.0, time = 140.5910.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Hybrid Simulations of the Global Venus/Solar Wind Interaction

Stephen H. Brecht, Principal Investigator
Berkeley Research Associates

Research Objective

The objective of this research is to obtain a better understanding of how the solar wind interacts with an unmagnetized planet such as Venus. Specifically we want to understand the dynamics of the shock and magnetic-barrier location as a function of solar wind parameters and ionospheric mass-loading rate.

Approach

We have developed a three-dimensional hybrid-particle code that simulates the electrons as a fluid and the ions as individual particles. Multiple species of ions are included, as is their complete kinetic behavior.

Accomplishment Description

The three-dimensional hybrid code called HALFSHEL was developed and is running on both Navier and Reynolds. We completed a study of the interaction of the solar wind with planets of various sizes. The sizes range from 1000 km to 6000 km in radius. These studies were performed with no ionosphere in the simulation. It was found, contrary to current thinking, that the shock and the magnetic barrier that forms to shield the planet surface are both asymmetric, without the presence of heavy ionospheric ions. It was found that the asymmetry is a function of the ratio of the planetary radius to the solar wind ion gyro-radius. In addition, it was seen that the plasma does not remain frozen to the magnetic field. A typical

simulation of a 6000-km-radius planet required 40 megawords of memory and about 16 hours of Cray Y-MP time.

Significance

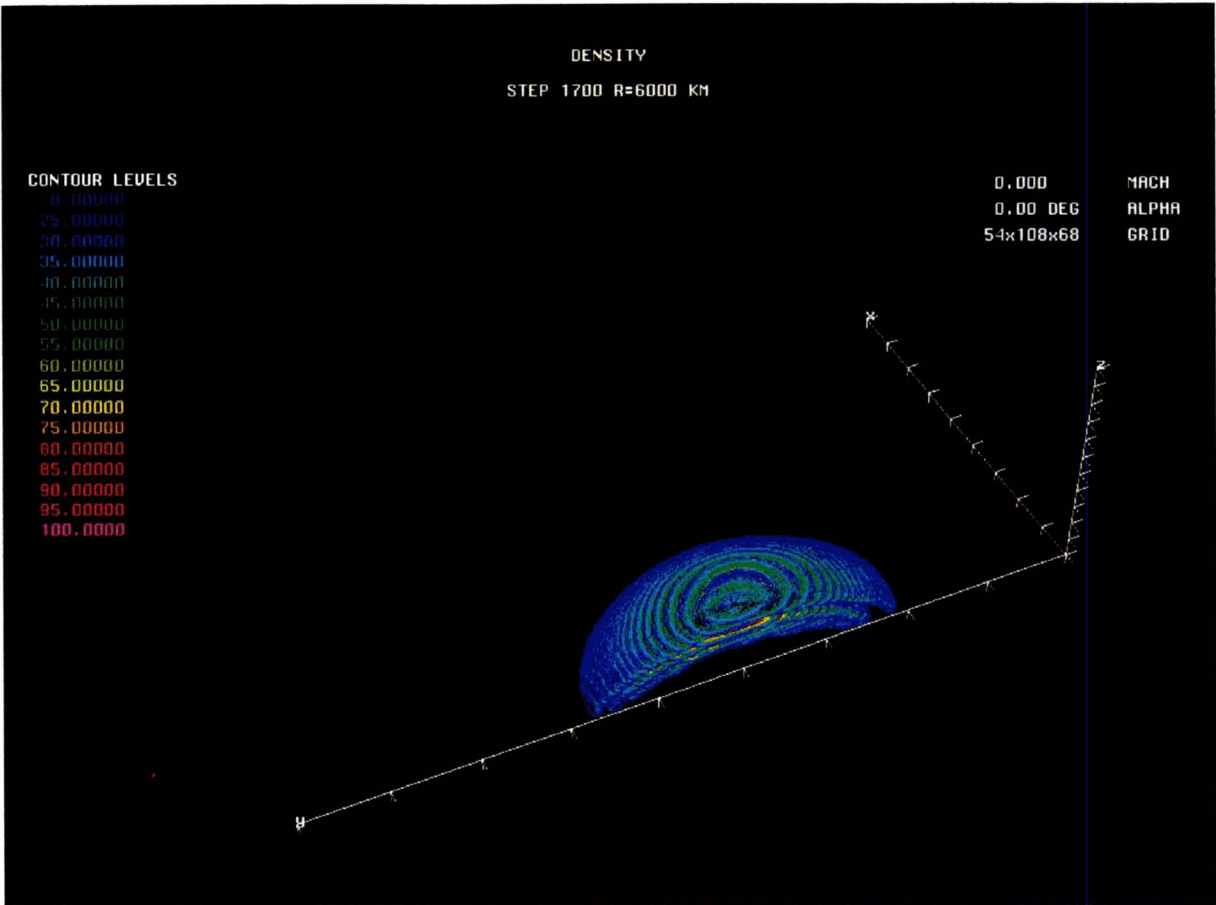
The results of these simulations and the visualization of these results are changing the way scientists view the interaction of the solar wind with planets such as Venus and Mars. These results are beginning to have an impact on how the data received from the Pioneer Venus Orbiter are interpreted. Specifically, the role of the ionospheric oxygen that is picked up during the interaction is being questioned.

Future Plans

The code is now being modified to include the ion mass loading produced by the charge exchange, ultraviolet ionization, and impact ionization of the oxygen in the Venusian atmosphere.

Publications

1. "Computer Simulation of the Solar Wind Interaction with Planetary Ionospheres." Invited paper, presented at the 6th International Association of Geomagnetism and Aeronomy, Exeter, England, July 1989.
2. "Magnetic Asymmetry of Unmagnetized Planets, EOS Transactions." *Amer. Geophys. Union* 70 (1989): 1176.
3. "Magnetic Asymmetries of Unmagnetized Planets." Submitted to *Geophys. Res. Lett.*, Feb. 1990.



Hybrid simulations of the global Venus/solar wind interaction.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Direct Simulation of Aerothermal Loads for the Aeroassist Flight Experiment

Edwin B. Brewer, Principal Investigator
NASA Marshall Space Flight Center

Research Objective

It is NASA's goal to develop computational codes to design the Aeroassisted Space Transfer Vehicle (ASTV). The Aeroassist Flight Experiment (AFE) will obtain the experimental data required to benchmark these codes for rarefied hypersonic flight with nonequilibrium reacting chemistry. The objective of the investigation described here is to develop a three-dimensional code applicable to the complex geometry of the AFE that can numerically determine the vehicle aerodynamics and aerothermodynamics in the transitional flow regime.

Approach

The computational approach used is the variable hard sphere (VHS) direct simulation Monte Carlo (DSMC) technique developed by G. A. Bird; a multispecies chemistry model is included. The DSMC approach is probably the only technique that can handle the complex physics involved.

Accomplishment Description

A three-dimensional body-fitted grid was generated for the AFE, including the carrier vehicle. The grid spacing, which varies with altitude, was adjusted to meet the size requirements relative to the local mean free path. The aerodynamics and aerothermodynamics of the AFE were obtained in the transitional flow regime from altitudes of 152 km down to

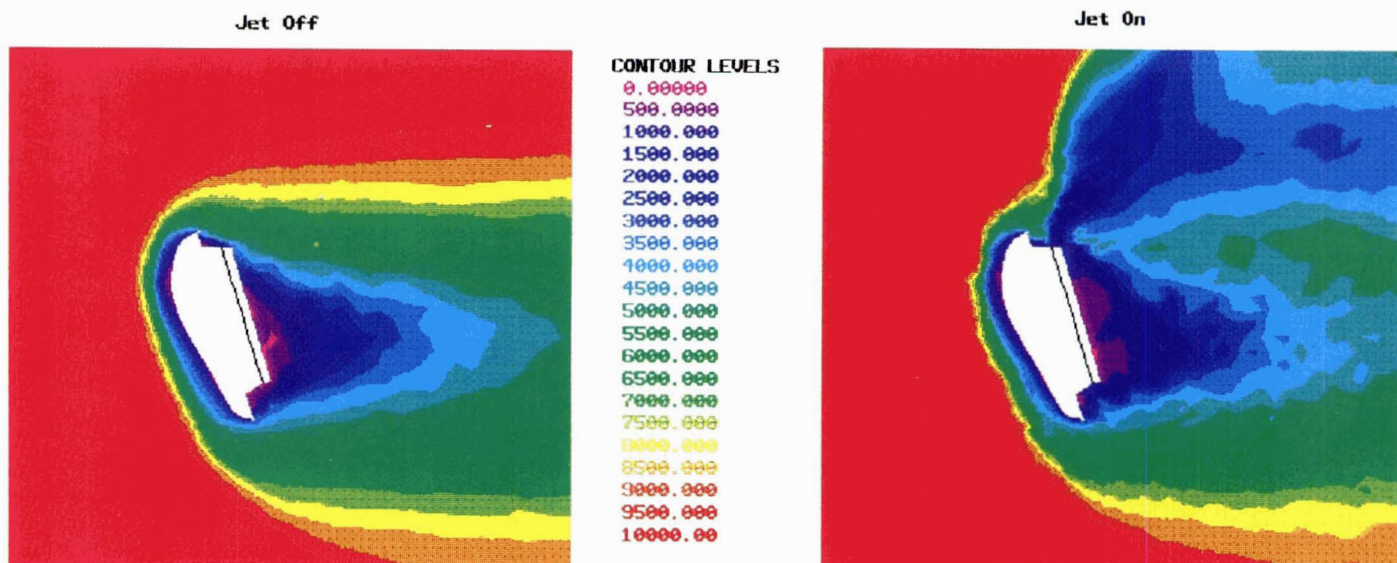
100 km, the lowest altitude that seemed practical using conventional DSMC. In addition, the flow field about the AFE, including the wake region, was determined. The approximate effects of a small reaction-control thruster was determined at the 100-km trajectory point. The accompanying figure shows the variation of velocity magnitude about the AFE with and without the thruster plume. The ammonia almost completely escapes the AFE wake, whereas a significant amount of hydrogen is caught up in the wake and base region. The 100-km computation required about 182 Cray-2 hours and 38 megawords of memory, with the simulation using about a million molecules and 80,000 computational cells.

Significance

It is expected that the ASTV will operate in the rarefied upper atmosphere in flight regimes that cannot be simulated in ground facilities. The AFE will furnish the benchmark data required to verify the ability of this code to predict ASTV design data in the transitional flow regime. In the meantime, the code is being used to verify the AFE design.

Future Plans

The effects of the control thrusters on the vehicle aerodynamics, the flow field, and the science experiments will be investigated during the NAS 1990-91 operational year.



Velocity magnitude about the AFE; altitude = 100 km.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Hypersonic Air-Breathing Missile Aero/Propulsion Integration

Adam C. Bricker, Principal Investigator
Co-investigators: Brad K. Bergman, Kurt Abrahamson, and Nancy Wilkins
General Dynamics, Convair Division

Research Objective

The objective of this work is to develop computational fluid dynamics (CFD)-based analytical methods applicable to the design of highly integrated, hypersonic air-breathing vehicles. Primary interest lies in the Mach 4 to Mach 6 speed regime. Complete configuration calculations including mass flowing inlets and nozzles are required. Specific developments required to accomplish this task are in turbulent flow modeling, blocked gridding concepts, flow-field grid adaptation, and bleed boundary conditions.

Approach

Three existing full Navier-Stokes (FNS) algorithms were investigated for accuracy and efficiency. Validations were made on various two- and three-dimensional hypersonic internal and external geometries. The three codes investigated were CFL3D (NASA Langley), PARC2D/3D (NASA Ames, AEDC), and FLOGD (General Dynamics Convair).

Accomplishment Description

A CFD-based method was developed for analyzing Mach 4 to Mach 6 class air-breathing vehicles. Three existing FNS algorithms were evaluated for a variety of internal- and external-flow cases. The codes vary substantially in their approaches to the solution of the Navier-Stokes equations, as indicated below.

- CFL3D: upwind flux-split, finite volume, implicit time marching, LU-ADI
- PARC2D/3D: central differenced, finite difference, implicit time marching, ADI

FLOGD: central differenced, finite volume, explicit time marching, Runge-Kutta

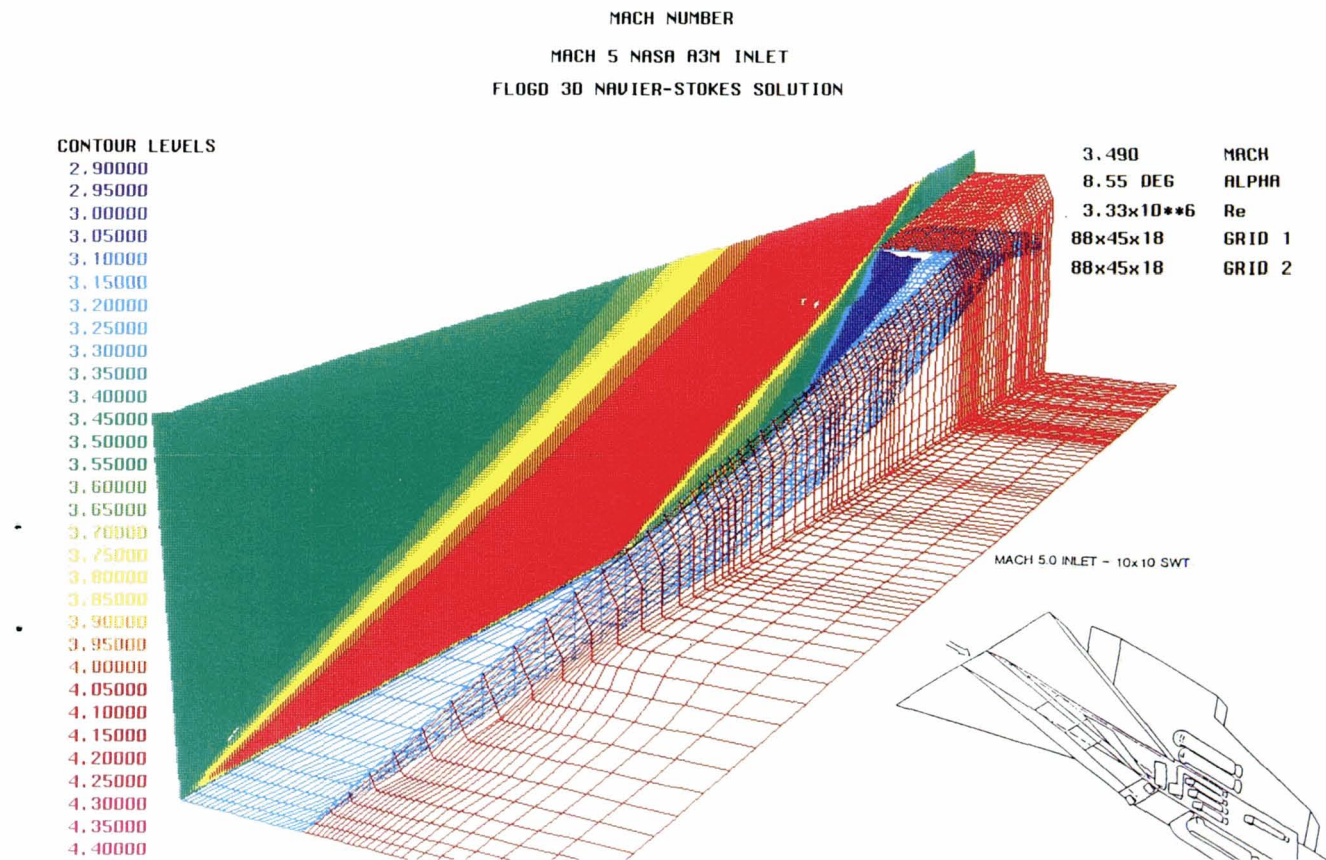
Conclusions drawn from this study indicate PARC and FLOGD solutions to be adequate for flows in this speed regime. Excellent correlation with existing test data was found for both codes. The shock-capturing ability of these central difference schemes was enhanced by an elliptic flow-field grid adaptation routine developed in-house. A sample FLOGD FNS solution for a recently tested NASA Lewis Mach 5 inlet is presented in the accompanying figure. The calculation required 6 Cray-2 hours and 31 megawords of memory. The flux-split upwind scheme used in CFL3D was found to be highly unstable for the cases run. Extremely low time steps were required to attain convergence, resulting in excessive run times. Researchers at NASA Langley are investigating reasons for this behavior and are presently investigating our test cases.

Significance

There has been increased emphasis on Mach 4 to Mach 6 class air-breathing vehicles. Presently there exists a void of aero/propulsion test data in this speed regime. The CFD tools provide an alternative source of data until testing results are made available. As in all situations, these tools require an initial development and validation phase.

Future Plans

Additional benchmarking as well as validation of the FLOGD and PARC codes is still required on more complex cases. Other tasks include the development of a two-equation turbulence model and investigation of multiprocessing capabilities.



A FLOGD, three-dimensional, full Navier-Stokes solution for a Mach 5 NASA A3M inlet.

Three-Dimensional Shear Layers and Wall-Bounded Compressible Turbulence

Jeffrey C. Buell, Principal Investigator
NASA Ames Research Center

Research Objective

To develop three-dimensional direct numerical simulation data bases and basic physical understanding of two flows: spatially developing, incompressible shear flows and turbulent wall-bounded compressible flows.

Approach

Direct numerical simulations of the full three-dimensional Navier-Stokes equations are used with resolution adequate to resolve all length scales. A mixed Fourier/finite-difference method is used in the incompressible shear flow code. Plane Couette flow was chosen as a special case of wall-bounded compressible turbulent flow and is studied using a new spectral code based on Fourier and Legendre polynomial expansions.

Accomplishment Description

Three-dimensional spatially developing mixing-layer calculations were carried out at a higher Reynolds number ($Re = 400$) than during the previous year ($Re = 200$). The new results show a more inviscid-like behavior during roll-up and pairing. The mixture fraction calculated from a "conserved" passive scalar and fast-chemistry assumptions is approximately midway between that of a temporal mixing layer (i.e., 0.5) and a two-dimensional spatial mixing layer. This implies that the longitudinal vortices tend to mix equal parts of the fast and slow streams and that the asymmetric entrainment of the spatial mixing layer is a two-dimensional effect. For the

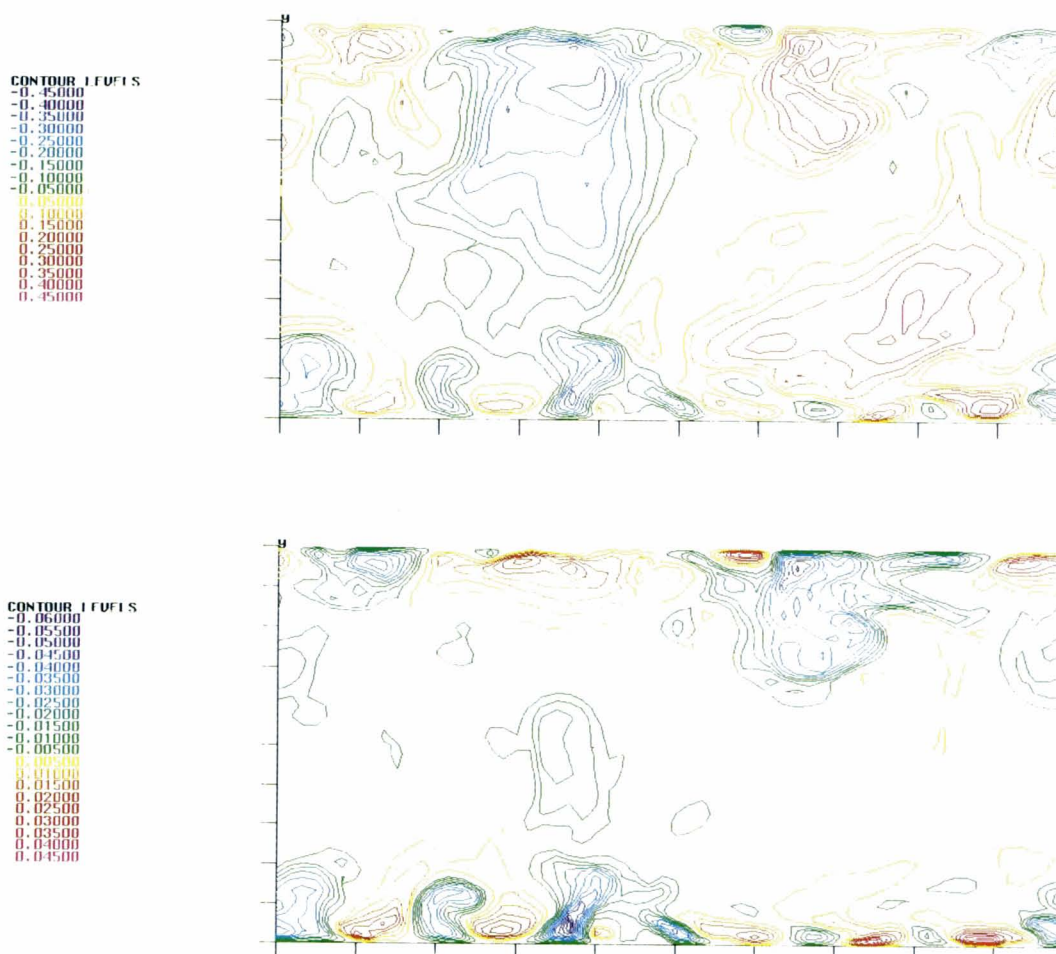
compressible wall-bounded turbulence project, a new spectral code was developed, and the first production runs were made by the end of the year. Fourier series are used in the horizontal directions and Legendre polynomials in the vertical direction. A partially implicit time-integration scheme is used that avoids time step restrictions that result from acoustic or diffusive effects. Shown in the figure are instantaneous streamwise velocity (top) and temperature (bottom) perturbations in a cross-stream plane at $M = 1$. In addition to the low- and high-speed streaks imbedded in the boundary layers, large eddies that scale on the channel width also form. On a typical $256 \times 144 \times 36$ grid, the incompressible free-shear code requires about 25 megawords of memory and 16 hours of Cray-2 CPU time per physical flow-through time. The compressible code requires about 50 hours of Cray Y-MP CPU time to obtain good statistics using $110 \times 64 \times 60$ modes.

Significance

The prediction of three-dimensional shear flows is part of many problems of engineering importance, especially ones related to combustion. Simulation data from the compressible Couette flow will be useful for supersonic boundary-layer turbulence modeling.

Future Plans

Different inflow boundary conditions will be studied using the incompressible shear flow code. Higher Mach numbers will be simulated with the compressible Couette code.



Instantaneous streamwise velocity (top) and temperature (bottom) perturbations in a cross-stream plane; $M = 1$.

Space Shuttle Flow Field

Pieter G. Buning, Principal Investigator

Co-investigators: J. L. Steger, R. L. Meakin, F. W. Martin, I.-T. Chiu, Y. M. Rizk, M. Yarrow, and S. Obayashi

NASA Ames Research Center

Research Objective

To develop the capability for accurate computational fluid dynamic (CFD) prediction of pressure loads on the Space Shuttle launch vehicle, and to explore flow-field simulation during such unsteady maneuvers as solid rocket booster (SRB) separation.

Approach

The three-dimensional thin-layer Navier-Stokes composite-grid Chimera/F3D code is used for the simulation of the flow about the Space Shuttle launch vehicle. For steady-state calculations, nine overlapped grids are used to represent the orbiter, external tank, solid rocket boosters with plume, and various pieces of attach hardware between the components. For unsteady SRB separation, three relatively coarse grids and 350,000 grid points are used. Approximately 12 hours of Cray Y-MP time are required for simulation of the first 0.68 seconds of SRB separation.

Accomplishment Description

During this year, wing-load increments resulting from the addition of SRB plumes were evaluated and shown to be comparable to wind tunnel-derived increments. Additional protuberances and pieces of attach hardware were modeled,

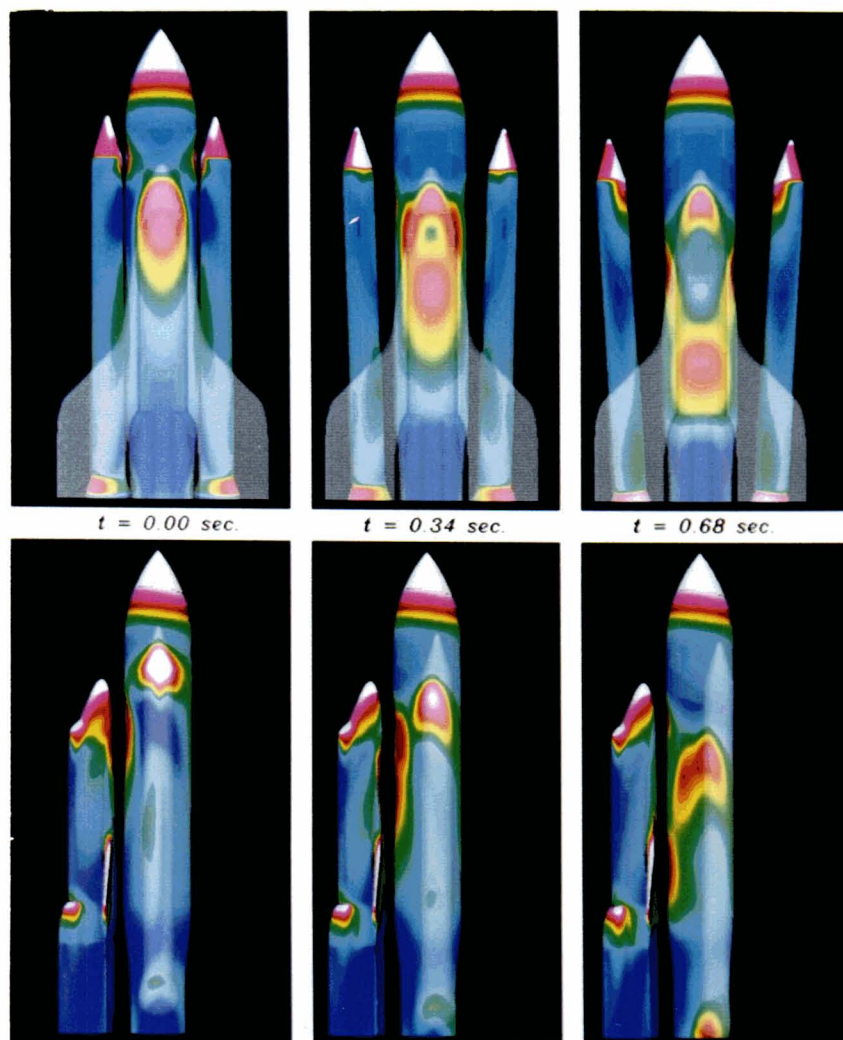
also showing significant effects on orbiter loads. A pioneering unsteady simulation of SRB separation at Mach 4.5 was made, showing the effect of SRB bow-shock impingement on the orbiter and external-tank surface pressures. Finally, the effect of acceleration on shock location and on the unsteady flow behind the external tank was studied.

Significance

Continuing uncertainties in orbiter structural loads limit payloads and affect gust-related launch constraints. Calculations resulting from this study are being used as a "third opinion" to add to wind tunnel and flight measurements, in an effort to resolve these uncertainties. In addition, the collection of flow calculations from Mach 0.6 through Mach 4.5 has served as a flow-field data base in analyzing the origin of excessive debris damage on STS-27.

Future Plans

The addition of real-gas effects for more accurate plume modeling is being undertaken, as is the development of techniques for handling pieces of geometry that intersect. New demonstration calculations will include Space Shuttle main engine plumes, flight Reynolds number, and return-to-launch-site abort flow fields.



ORIGINAL PAGE
COLOR PHOTOGRAPH

Two-Body Hypersonic Separation Analysis

Geoffrey Butler, Principal Investigator

Co-investigators: Brian Nguyen, Eric Sharnhorst, and David King

General Dynamics, Convair Division

Research Objective

This research studies the practical problems of store separation from hypersonic vehicles. The ultimate goal is to develop analysis procedures for the prediction of multiple bodies separating at high speed. The intermediate steps involve the solution of hypersonic flow fields through the use of computational fluid dynamics (CFD) techniques.

Approach

An approximate method using quasi-steady assumptions, and a completely unsteady method, are considered. In the approximate method, a single CFD solution of the hypersonic flow field about a carrier vehicle is obtained (see figure), and a multiple-body, six-degree-of-freedom (6DOF) simulation program is used to model the trajectory of a submissile flying through the flow field. The unsteady simulation method solves the flow equations and the 6DOF rigid-body equations simultaneously. The unsteady flow equations are solved on an unstructured grid that evolves with the problem's changing geometry.

Accomplishment Description

The implicit finite-difference code PARC was used to generate axisymmetric solutions for the body and wake of a generic hypersonic vehicle. Since wake solutions using PARC have not been thoroughly validated, these solutions were converted to interferogram fringes and compared to laser interferograms of similar vehicles in free flight at the AFATL/FXA Aeroballistic Range Facility. Good general agreement was obtained between the CFD solutions and the test data. The CFD solutions were then used to predict the trajectories of submissiles ejected into the wake of the generic carrier. Two- and three-

dimensional unstructured-grid generators were developed and used on arbitrary, multiple-body configurations. A two-dimensional unstructured-grid flow solver was developed and run, giving preliminary results. The cases so far run needed about 10 minutes and less than 1 megaword on the Cray Y-MP.

Significance

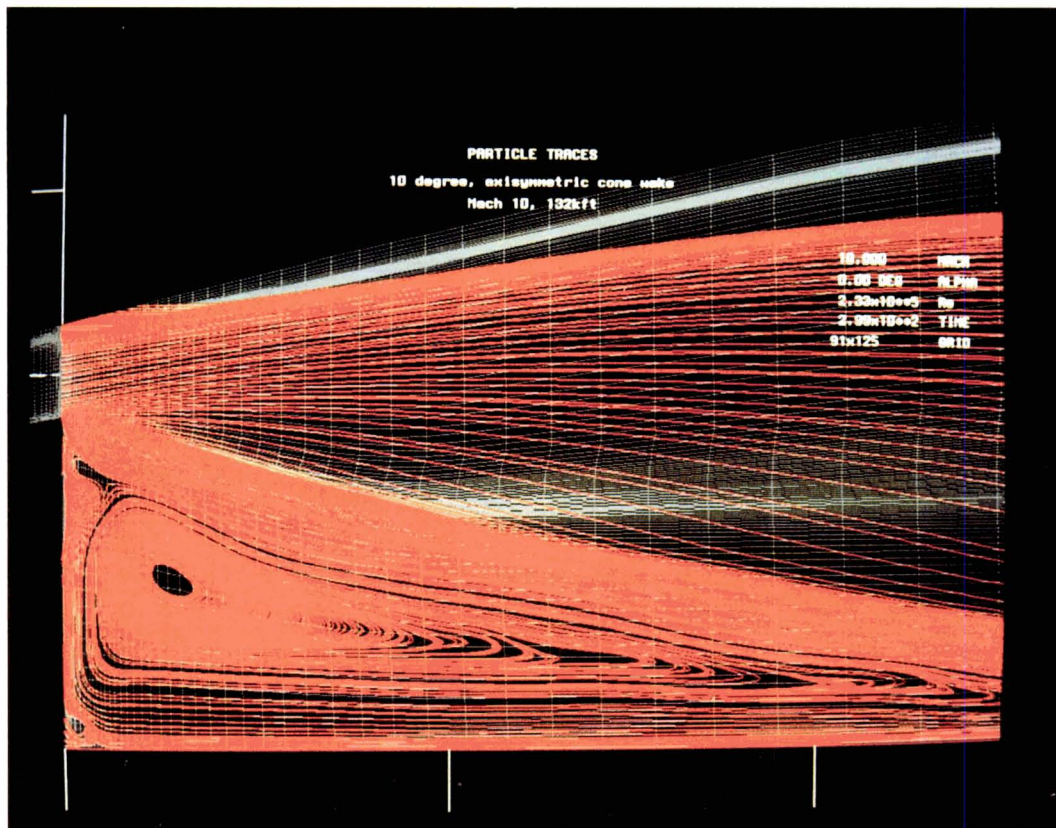
Vehicle separation during high-speed flight is seen in weapon/external-tank separation systems for supersonic and hypersonic platforms, in crew escape modules from the National Aero-Space Plane (NASP) or other high-speed vehicles, and in the stage separation of launch vehicles. Such separations can be extremely hazardous, but existing ground-based facilities are unable to simulate multiple bodies at release flight conditions. This research is dedicated to the development of a complete simulation for case studies of store separation as well as a less costly simulation that can be used to perform preliminary studies.

Future Plans

The flow solver will be expanded to three dimensions and to include viscous terms and real gas effects. The grid generator and flow solver will be combined to adapt the grid to gradients in the solution. Moving boundaries and moving grid points will be incorporated. Finally the codes will be used to give time-accurate solutions of multiple moving bodies.

Publications

Butler, G.; King, D.; Nguyen, B.; and Abate, G. "Ballistic Range Flowfield Measurements of the Hypervelocity Near Wake of Generic Shapes and Correlation with CFD Simulations." AIAA Paper 90-0621, Jan. 1990.



The hypersonic flow field about a carrier vehicle.

Receptivity, Eigenfunction Modeling, and Simulation of a Wall-Bounded Flow

Alan B. Cain, Principal Investigator

Co-investigators: William W. Bower and Nagi N. Mansour

McDonnell Douglas Research Laboratories/NASA Ames Research Center

Research Objective

To better and more usefully characterize transition and turbulence.

Approach

One recent effort of this study focused on the use of a fractional step code developed by Nagi N. Mansour and Jeffery C. Buell of NASA Ames, to investigate the flow over a backward-facing step. A separate effort utilized a McDonnell Douglas Research Laboratories (MDRL) spectral code to simulate an excited free shear flow.

Accomplishment Description

The Mansour-Buell code was used to investigate the flow over the backward-facing step. Recent results of subharmonic and fundamental forcing show a close correspondence to MDRL experimental results and confirm some speculation on the nature of the unsteady reattachment process. The MDRL spectral code simulations have identified key mechanisms that also explain experimental observations for free shear flows. The accompanying figure illustrates the vorticity contours for the backward-facing step at $t = 164$ sec in the unsteady simulation for a Reynolds number of 2000, based on step height. A grid containing 201 points in the streamwise

direction and 65 in the normal was used in the simulation, which required slightly more than an hour of CPU time.

Significance

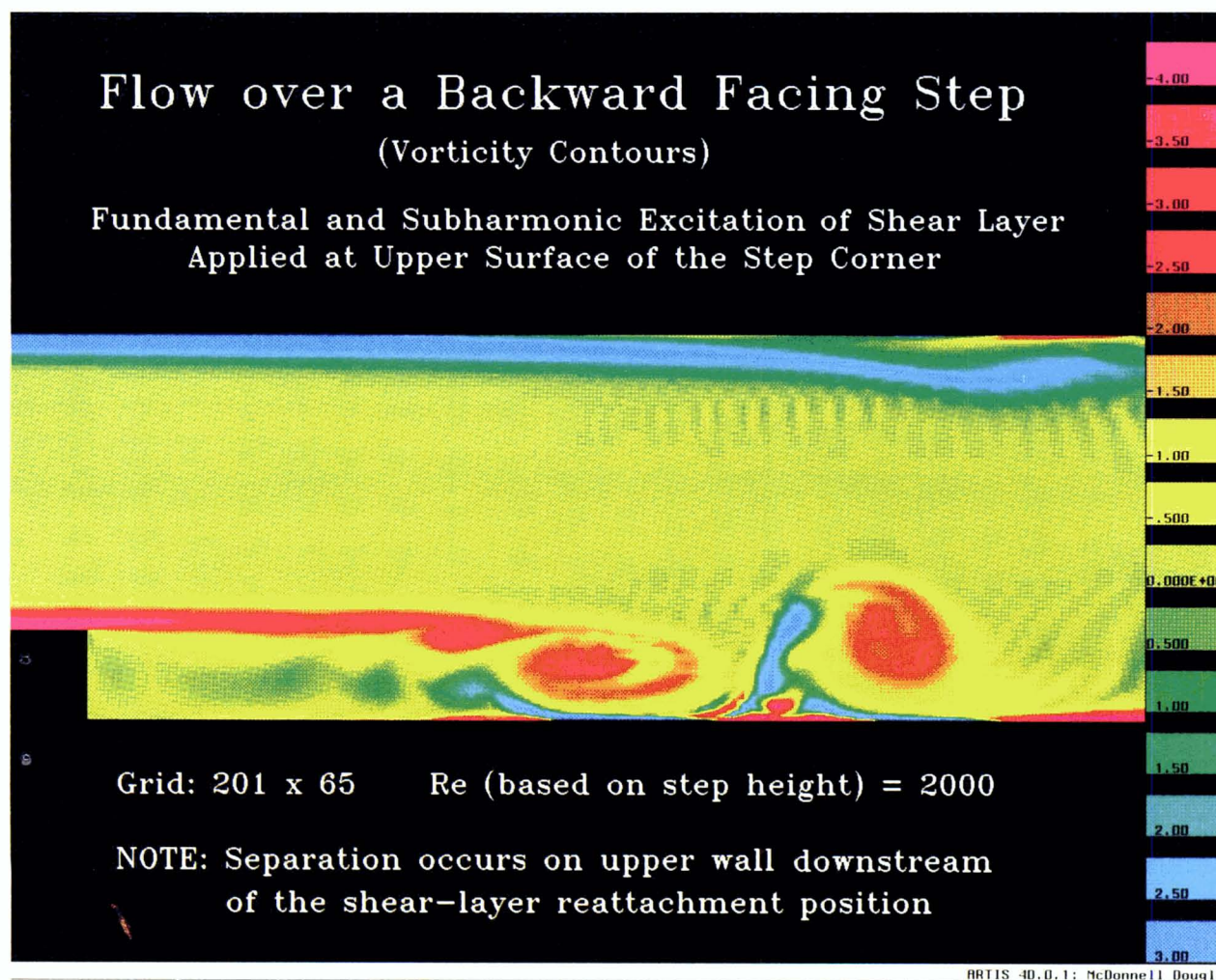
The results of this basic study are important to the beneficial use of techniques for actively controlling shear flows. These results also provide the insight and motivation to try new approaches to the modeling of large-scale turbulence in these flows.

Future Plans

We are branching into new directions from our initial work. The extended effort is focusing on linear, nonlinear, incompressible, and compressible formulations to examine the behavior of receptivity and transition. The fully simulated calculations will be compared with simplified models of the flow fields.

Publications

1. Cain, A. B.; Roos, F. W.; and Kegelmann, J. T. "Computational and Experimental Observations on Modal Characteristics of an Excited Free-Shear Layer." Presented at the Twelfth Symposium on Turbulence, Univ. of Missouri-Rolla, Oct. 1990.
2. Cain, A. B.; Mansour, N. N.; Buell, J. C.; and Bower, W. W. "Simulations of Flow over a Backward-Facing Step with Forcing." To be submitted.



An Unstructured Triangular-Mesh/Navier-Stokes Method for Computing the Aerodynamics of Aircraft with Ice Accretion

Steven C. Caruso, Principal Investigator
Nielsen Engineering & Research, Inc.

Research Objective

The long-range objective of this work is to develop a flow-field prediction method that can be used for the analysis of complex, time-dependent, ice accretion shapes on three-dimensional aircraft components and full configurations.

Approach

An unstructured, triangular-mesh/Navier-Stokes prediction method will be developed to compute flow fields about airfoils with quasi-time-dependent leading-edge ice accretions.

Accomplishment Description

A new method was developed to permit efficient mesh generation about airfoils with time-dependent leading-edge ice accretions. Unstructured meshes are generated about quasi-steady, iced airfoil surfaces using an approach in which only a small portion of the mesh in the vicinity of the new ice surface must be regenerated at each time step. Steady, unstructured-mesh Euler flow-field computations were made for clean and iced airfoils. Similar calculations were also made with a structured-mesh code. The results show that numerical accuracies are comparable; however, the unstructured-mesh CPU times are lower because of the efficient use of grid

points. More importantly, the new unstructured-mesh approach requires significantly fewer man-hours to generate a mesh about a complicated leading-edge ice shape (see figure). A single flow-field calculation requires about 10 minutes and 2 megawords on the Cray-2. A total of 16 Cray-2 CPU hours were used in the course of this study.

Significance

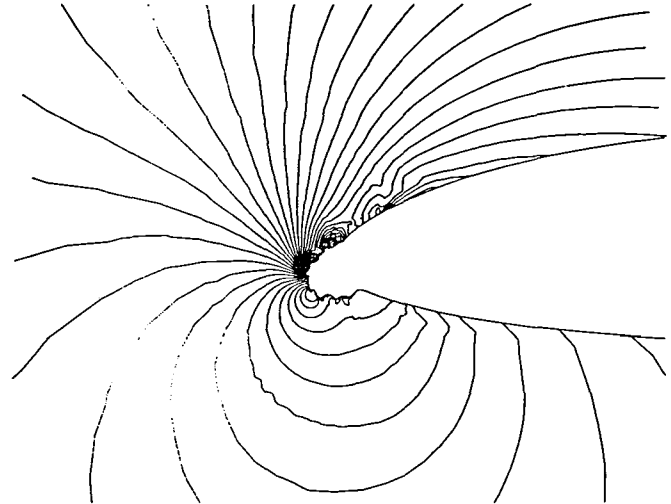
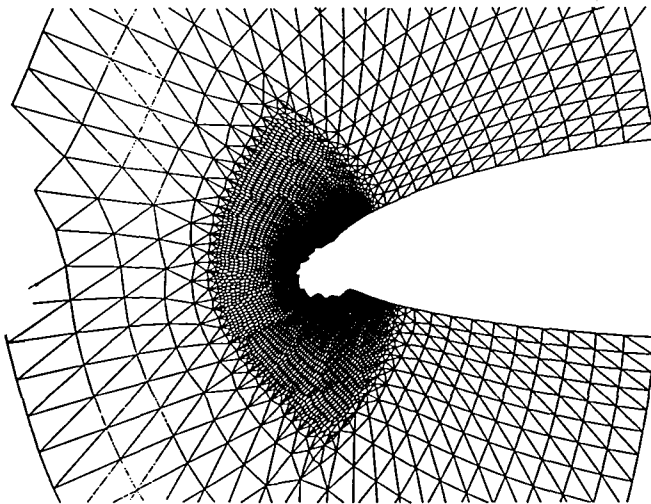
The results of the study demonstrate that the unstructured-mesh approach is a feasible flow-field prediction method for use in icing analyses. Further development is required before it can be used as a practical engineering tool.

Future Plans

The flow solver will be extended to the Navier-Stokes equations, with a suitable turbulence model. The grid generation method will be further automated to permit time-dependent icing analyses. Subsequent development will include extension of the method to three dimensions.

Publications

Caruso, S. "Development of an Unstructured Mesh/Navier-Stokes Method for Aerodynamics of Aircraft with Ice Accretions." AIAA Paper 90-0758, 1990.



Flow-field computation for airfoil with simulated leading-edge ice accretion. (Left) Mesh near leading edge. (Right) Normalized pressures near leading edge.

Fuel/Air Turbulent Combustion in a Three-Dimensional Combustor Geometry in Supersonic Flow

Sukumar R. Chakravarthy, Principal Investigator

Co-investigators: Uriel C. Goldberg and Sampath Palaniswamy
Rockwell International Science Center

Research Objective

To understand the details of supersonic mixing and its influence on the stability of the combustion process.

Approach

The unified solution algorithms (USA) series codes are highly suited for simulating turbulent supersonic reacting flows. They use a second-order-accurate (in space) finite-volume, multi-zone, total-variation-diminishing upwind scheme to solve compressible, chemically reacting flows. This scheme avoids spurious numerical oscillations without adding an artificial dissipation term to the discretized equations. It has an algebraic model (a modified Baldwin-Lomax model with Goldberg's shear-layer treatment away from wall boundary layers), a one-equation model (a partial differential equation for the turbulent kinetic energy), and a two-equation model (partial differential equations for turbulent kinetic energy and turbulence dissipation) to solve the Reynolds-averaged Navier-Stokes equations. All models incorporate Goldberg's backflow model to treat recirculatory flow regions.

Accomplishment Description

Experimental work done at the University of Virginia with tangential injection of fuel in a supersonic airstream was chosen as one of the case studies since it was designed to enhance mixing using vortical main flow. Using NAS resources we have obtained a three-dimensional solution to the mixing problem (nonreacting flow). The algebraic turbulence model available in the USA code was used and the flow was computed using a four-zone grid with 67,000 grid points. Comparison with preliminary experimental data shows that we can predict the splitting of the fuel jet caused by the swirling throughflow, as well as the tracer concentration (see figure). Fifty Cray Y-MP hours and 8 megawords of CPU memory were required for this computation.

Significance

An essential part of a hypersonic flight vehicle is an engine in which fuel can be burned efficiently in a supersonic airstream. Constraints on the weight and the size of the combustion chamber require that the fuel-air interactions be optimized to enhance mixing and that the geometry be selected to stabilize the flame within the chamber. Since the airflow inside the scramjet engine is supersonic, the goal is to achieve mixing with minimal losses and complete combustion of the fuel. Mixing in the shear layer can be increased if vortical flows can be established in the fuel and/or airstreams. Computational tools that can accurately predict the mixing rate are critical to scramjet engine development. The numerical algorithm should have high-order accuracy to avoid smearing and numerical mixing. This requires the numerical scheme to have minimal dissipation even while resolving flows with strong shock waves and shock-wave boundary-layer interactions. The turbulence models used in the Reynolds-averaged Navier-Stokes equations should faithfully reproduce the effect of turbulent mass, momentum, and energy transport on the mean flow.

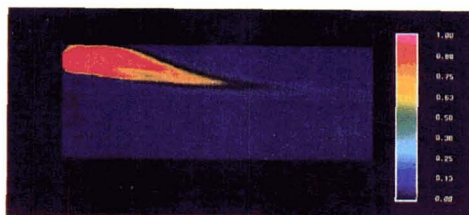
Future Plans

We plan to direct our efforts on two fronts. On the first front we plan to (a) test and validate our turbulence models for three-dimensional supersonic flows with multispecies mixing and chemical reactions, (b) improve existing and develop new turbulence models for these flows, and (c) quantify and predict mixing rates. These tasks will be performed using the geometry proposed by researchers at the University of Virginia. On the other front, parallel with the first one, we propose to study various geometries and predict optimal configurations for secondary flow-induced mixing enhancements for scramjet applications. We have begun looking at swept ramps (delta wing forms) as one possibility.

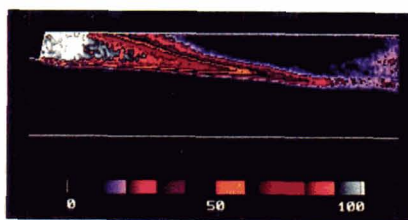
CFD RESULTS SHOW GOOD AGREEMENT WITH UNIVERSITY OF VIRGINIA TEST DATA (U)

3-D, FNS, FC, TR CALCULATIONS — 67,045 GRID POINTS

CFD TRACER CONCENTRATION PREDICTIONS
Y/D = 0



UNIVERSITY OF VIRGINIA PLIF DATA
Y/D = 0



- SUBSCALE SINGLE INJECTOR (10° RAMP) TESTED AT $M = 2$ AT UNIVERSITY OF VIRGINIA
- IODINE BASED PLIF MEASUREMENTS SHOW BIFURCATION OF JET SEEN IN COLD FLOW CFD SIMULATIONS
- RESULTS SHOW DESIRED VORTEX INDUCED MIXINGS CHARACTERISTICS

ORIGINAL PAGE
COLOR PHOTOGRAPH

Fuel/air turbulent combustion in a three-dimensional combustor geometry in supersonic flow.

Three-Dimensional Calculations of Multiple Control-Jet Interactions for Defense Interceptors

R. Rexford Chamberlain, Principal Investigator
Lockheed Missiles and Space Company, Inc.

Research Objective

The present objective is to develop the ability to predict the influence of multiple control-jet-interaction phenomena on the forces and moments experienced by hypersonic defense interceptors. As a first step, numerical simulations of a single supersonic jet exhausting into hypersonic crossflow at various jet exit pressures, angles of attack, and Mach numbers are considered.

Approach

A three-dimensional upwind Navier-Stokes code is used to compute the jet interaction flow field about a generic interceptor configuration. Predicted surface pressures are compared with wind tunnel data, and forces and moments are calculated.

Accomplishment Description

An implicit, upwind, finite-volume Navier-Stokes code was developed and validated for both two- and three-dimensional applications. The inviscid fluxes were treated using either flux-vector (van Leer) or flux-difference-splitting (Roe) techniques so that the complicated shock systems that result from the jet interaction could be crisply resolved. The full viscous terms were included using central differencing throughout. Calculations at various jet exit pressures and angles of attack were used to examine the nature of the jet interactions over a generic configuration. The accompanying graphic shows the pressure contours over the vehicle at Mach 8 and 0° angle of attack. The jet plume extends beyond the body bow shock and leaves a trace of high pressure along the rearward portion of the surface (flare). The interaction accounts for about a 10%

increase in the thrust of the jet, and the distribution of the force is such that the pitching moment is also amplified. This calculation required about 15 Cray-2 hours and 10 megawords of memory.

Significance

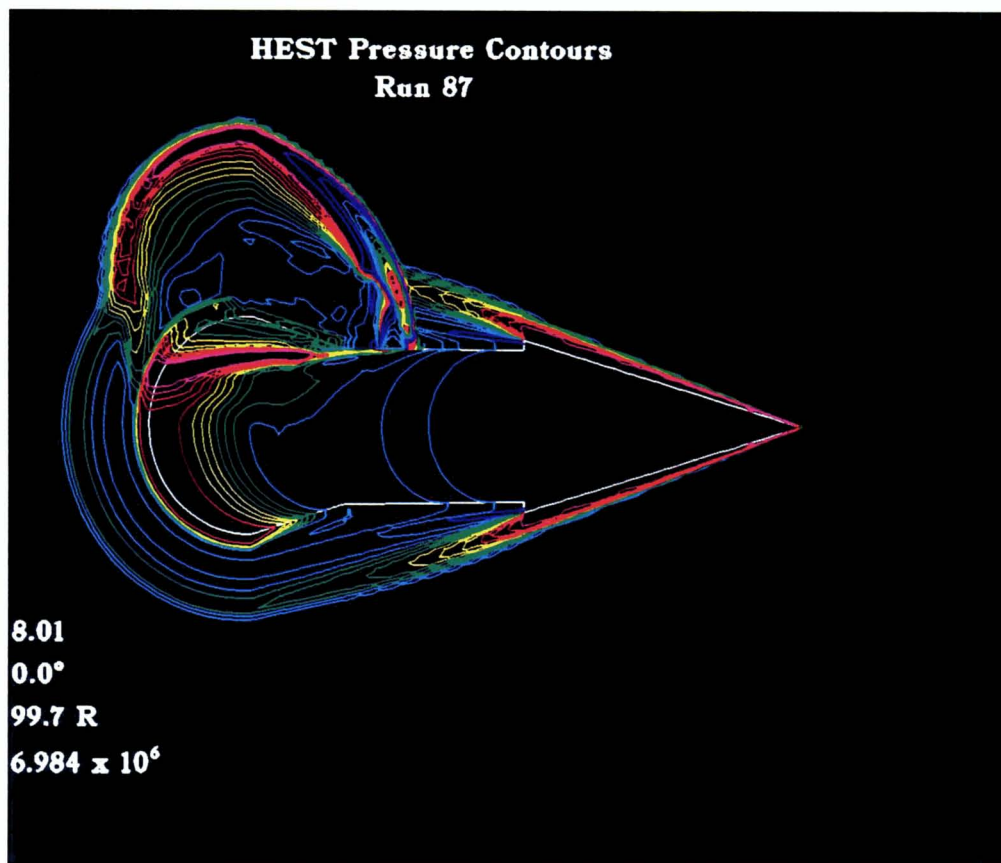
For future- and current-generation defense interceptors, designers are examining the head-end steering concept, in which a single reaction-control system located near the nose of the vehicle is responsible for maneuvering during the boost phase as well as the end game. The evaluation of this concept will depend on a detailed understanding of the fluid dynamics associated with control-jet interactions as well as on predictions of the control effectiveness. Numerical simulations will contribute significantly to the performance evaluation of proposed interceptor configurations under the desired operating conditions. In addition, the creation of a simulation data base will enhance the intuitive understanding that is required to make critical design decisions.

Future Plans

The existing code will be used to continue the study of the complex flow structures associated with the interaction of single and multiple control jets. A natural extension of the present method is to incorporate a turbulence model that is fully consistent with the upwind treatment of the convective transport terms in the finite volume setting.

Publications

Chamberlain, R. R. "Calculation of Three-Dimensional Jet Interaction Flowfields." AIAA Paper 90-2099, July 1990.



HEST pressure contours, run 87; $M_\infty = 8.01$, $\alpha = 0.0^\circ$, $Re = 6.984 \times 10^6$, $T_\infty = 99.7R$.

Continuum Computational Fluid Dynamics for Hypersonic High-Altitude Flight

Dean R. Chapman, Principal Investigator
Co-investigator: Robert W. MacCormack
Stanford University

Research Objective

To develop advanced continuum equations of gas motion that are more physically realistic than the conventional Navier-Stokes equations for hypersonic flows at high altitude, and to develop numerical solution methods for the advanced set of equations.

Approach

Modifications of the Burnett equations are explored for adequately representing the strongly nonequilibrium flow conditions that exist in hypersonic shock-wave structures at high altitudes. Both experimental data and direct simulation Monte Carlo (DSMC) computations are used for test purposes.

Accomplishment Description

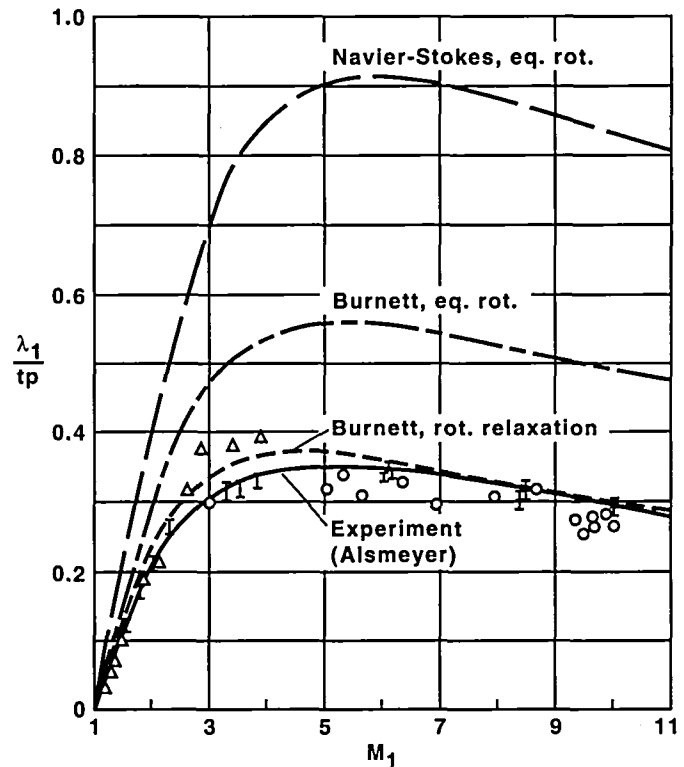
The Burnett equations were recently found to be much more accurate than the Navier-Stokes equations for computing one-dimensional hypersonic shock structure in monatomic gases and nitrogen (see accompanying graph). In the latter case, a rotational energy relaxation model must be added to the Burnett equations. Solutions of the Burnett equations for planar two-dimensional flow fields were recently undertaken. A fundamental problem of Burnett instability in regard to flow perturbations of very small wavelengths was encountered and overcome in one-dimensional flow by the formulation of "augmented" Burnett equations. The Burnett equations for axially symmetric flow were derived for the first time, although no numerical solutions have yet been attempted. Preliminary coarse-mesh computations of planar two-dimensional flow indicates that the CPU time required for Burnett solutions is about 40% greater than that required for corresponding Navier-Stokes solutions. An average job run would need about 1.5 hours and 5 megawords of memory.

Significance

Vehicles such as the Aeroassisted Space Transfer Vehicle and the Aeroassist Flight Experiment, which use atmosphere braking for orbital changes, operate at altitudes well above the range where the conventional Navier-Stokes equations are accurate. Ground-based experimental facilities are incapable of reproducing the combined flow parameters necessary for simulating these nonequilibrium hypersonic flow conditions. Therefore, advanced continuum equation sets, such as the Burnett equations, are needed for the efficient and accurate design of these particular hypersonic vehicles. Other relevant applications involve the flow over small-radius cowl lips and leading edges on the National Aero-Space Plane.

Future Plans

Solutions will be attempted to the full Burnett equations for axially-symmetric flow as well as two-dimensional planar flow. Once successful, three-dimensional solutions will be attempted.



Reciprocal shock-wave thickness in nitrogen.

Hypersonic Flow Past Generic Lifting Bodies

Denny S. Chaussee, Principal Investigator

Co-investigators: Scott L. Lawrence, Thomas A. Edwards, and Bradford Bennett

NASA Ames Research Center

Research Objective

The ultimate goal of this effort is the development of the capability to compute the entire aerothermal environment of a vehicle such as the National Aero-Space Plane (NASP) in hypersonic flight. The computations include the external flow as well as the flow into the inlet, through the combustor, and out the nozzle—the so-called tip-to-tail capability.

Approach

The CNS and UPS codes solve the Navier-Stokes and parabolized Navier-Stokes equations, respectively, using algorithms appropriate for high Mach number flows. Because of its speed and relative efficiency, the UPS code is used for streamlined regions of a configuration. The zonal method used by the CNS code makes the CNS code well suited for complex portions of the vehicle, such as inlets, wings, and fins. Both codes include turbulence, transition, and equilibrium and finite-rate air chemistry models.

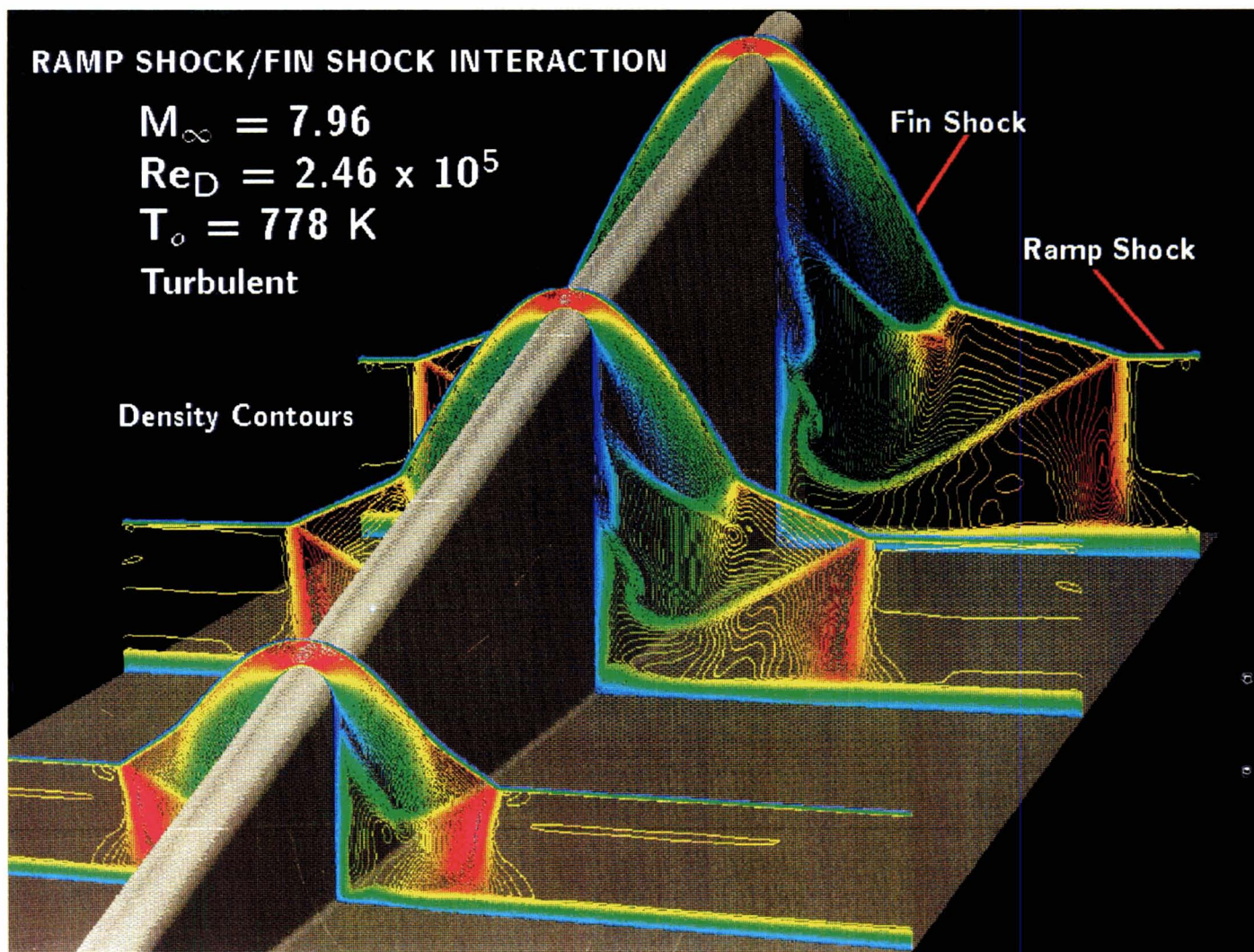
Accomplishment Description

Validation has begun for the nonequilibrium air chemistry models implemented in the CNS and UPS codes. A version of

CNS was created for the calculation of hypersonic internal flows, and this version is in the process of validation using a side-wall compression inlet case for which experimental pressure and pitot pressure data exist. The UPS code was used in the investigation of the kind of shock-shock interaction that might occur where a bow shock impinges on a wing or tail (see accompanying figure). A highly refined grid was necessary to resolve the details of the interaction. In order to provide the same type of resolution with fewer grid points, a solution-adaptive gridding procedure is being implemented within the UPS code.

Significance

Because ground-based facilities are unable to simulate the environment that the NASP will encounter, it has been determined that the NASP program needs the full support of computational fluid dynamics research to design a flight vehicle. The current effort has been undertaken to develop technology for use by the contractors in the design process as well as by the government in the assessment of proposed designs.



Turbulent ramp-shock/fin-shock interaction; $M_{\infty} = 7.96$, $Re_D = 2.46 \times 10^5$, $T_0 = 778 \text{ K}$.

Calculation of Transonic and Supersonic Flows Using an Interactive Scheme Based on Euler and Boundary-Layer Equations

Lee T. Chen, Principal Investigator

Co-investigators: Minh H. Bui and Hsun H. Chen

Douglas Aircraft Company

Research Objective

To develop and validate an interactive scheme based on Euler and boundary-layer equations, and to apply the scheme to calculate solutions about generic National Aero-Space Plane (NASP) vehicles and subsonic/supersonic transport aircraft.

Approach

The Euler method is based on Jameson's cell-centered and cell-vertex schemes, and the boundary-layer method is based on Cebeci's inverse-boundary-layer method. The Euler method was modified for various grid topologies so that it could be applied to vehicles with different body-nose and wing-tip geometries. The boundary-layer calculation was performed along prescribed approximate streamlines.

Accomplishment Description

The present scheme was applied to calculate transonic and supersonic flow solutions about generic NASP configurations and a Douglas AST wind tunnel model. A series of surface solutions obtained using H-O and C-C-C grid topologies were compared. The comparison indicated the importance of the

mesh size and the choice of grid topologies. Typical runs based on grid systems with 120,000 to 350,000 grid points required about 5 to 20 megawords of memory and used about 5 minutes to 1 hour of Cray-2 CPU time.

Significance

The Euler/inverse-boundary-layer method was successfully applied to calculate transonic solutions for various configurations including the NASP vehicle. The integrated force and moment components agree with test data.

Future Plans

The present method will be further used to validate solutions for generic NASP, AST, and subsonic transport configurations. More accurate streamlines will be prescribed for the boundary-layer calculation.

Publications

Chen, Lee T., and Bui, Minh N. "An Interactive Scheme for Transonic Wing/Body Flows Based on Euler and Inverse Boundary-Layer Equations." AIAA Paper 90-1586, June 1990.

DOUGLAS AST at $M=0.9$, and $\alpha=5^\circ$.



ARTIS: McDonnell Douglas Corp.

Flow about a Douglas AST wind tunnel model; $M = 0.9$, $\alpha = 5^\circ$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Airframe/Inlet Aerodynamics

W. J. Chyu, Principal Investigator
NASA Ames Research Center

Research Objective

To develop efficient numerical techniques for the analysis of inlets on highly maneuverable aircraft.

Approach

A compressible Navier-Stokes solver capable of handling a multiblock grid system is used to compute integrated external and internal flow fields.

Accomplishment Description

Computational capabilities for handling highly three-dimensional inlet geometries are being developed by using a multiblock, modified version of the implicit, partially flux-split, Navier-Stokes code F-3D. Flow fields are computed over an offset super-elliptic subsonic diffuser inlet (cross sections varying from rectangular to circular) with aspect ratio 4.0 and an offset of one exit diameter. The computed result compares well with the measured static pressures on the inlet wall and the total pressures on the engine face. It also reveals flow

features of a complex inlet flow field involving separated boundary layers and streamwise vortices. The solution takes about 32 Cray-2 hours and uses less than 15 megawords of central memory for the flow field containing 874,000 computational grid points.

Significance

The increased maneuvering requirements of advanced aircraft dictate much more complicated inlet geometries than are currently being used. The current level of computational techniques permits analysis of these more complex inlets, making possible the investigation of design variables associated with integration of the forebody/inlet system.

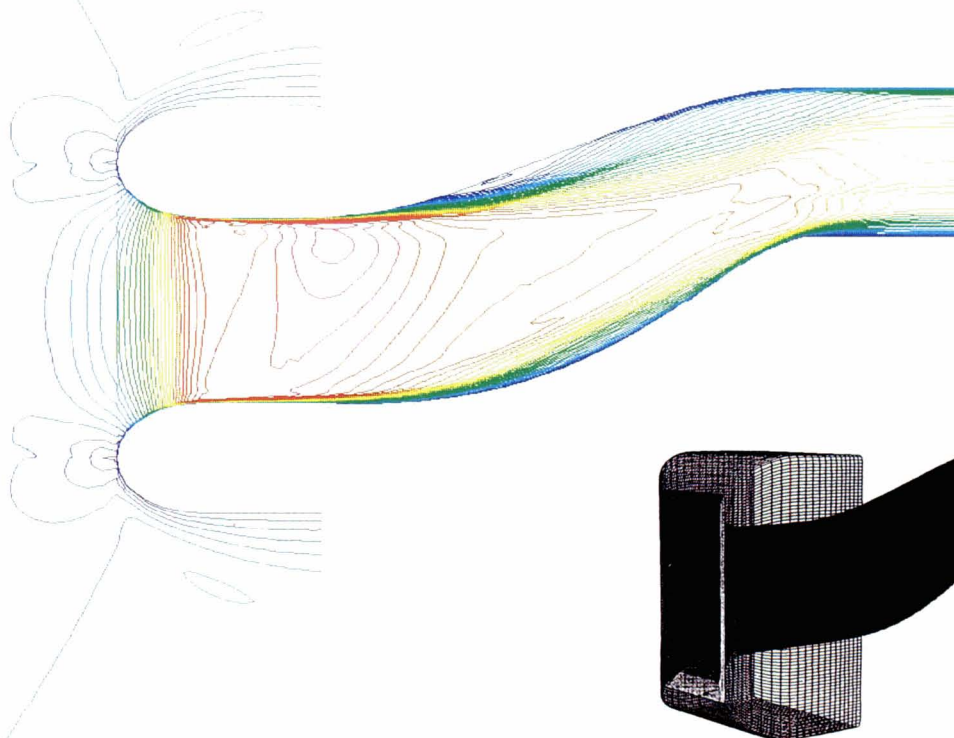
Future Plans

Future efforts will focus on the code validation cases for these unconventional inlet geometries, including the integration of internal and forebody solutions.

Mach Contours

CONTOUR LEVELS

0.00000
0.02000
0.04000
0.06000
0.08000
0.10000
0.12000
0.14000
0.16000
0.18000
0.20000
0.22000
0.24000
0.26000
0.28000
0.30000
0.32000
0.34000
0.36000
0.38000
0.40000
0.42000
0.44000
0.46000
0.48000
0.50000
0.52000
0.54000
0.56000
0.58000
0.60000
0.62000
0.64000
0.66000



Airframe/inlet aerodynamics; Mach contours.

Three-Dimensional Shock/Boundary-Layer Interactions in Hypersonic Inlets

William J. Coirier, Principal Investigator
NASA Lewis Research Center

Research Objective

The goal of this research is to investigate and characterize three-dimensional shock/boundary-layer interaction phenomena in rectangular supersonic and hypersonic inlets. The glancing shock/boundary-layer interactions in these inlets can cause highly distorted profiles at the inlet face, which in turn can drastically impair the inlet performance. A detailed understanding of the genesis of these distortions may help in the elimination or reduction of their effect on the performance of the entire vehicle.

Approach

An existing three-dimensional Reynolds-averaged Navier-Stokes code is modified to improve its modeling of high-enthalpy, high-Mach number flows, by including a high-order monotone upwinding scheme and by implementing an efficient approach to account for real gas effects. The original implicit algorithm in the code is replaced by a more robust and efficient implicit scheme for solving the three-dimensional Navier-Stokes equations.

Accomplishment Description

The original second-order central-differencing scheme of the code was replaced by a high-order monotone upwinding scheme that was based on flux vector splitting. Monotonicity was achieved by using either a MUSCL primitives extrapolation approach or a flux extrapolation approach. Real gas effects were included using a generalized equation of state, which was shown to be very efficient by allowing complete vectorization without restructuring of the code, and was shown to be flexible by allowing different equations of state to be easily supplied to the code. The original diagonalized Beam-Warming implicit scheme was replaced by the lower-upper symmetric Gauss-Seidel scheme, which was shown to be

efficient and robust for solving the three-dimensional Navier-Stokes equations with real gas effects. The improved code was then validated against experimental data for the hypersonic flow through a corner geometry, and was used to investigate the aerodynamic and heat transfer loads on a simulated scramjet inlet gap-seal system. Typical calculations used approximately 10 Cray-2 single processor hours and approximately 8 megawords of memory.

Significance

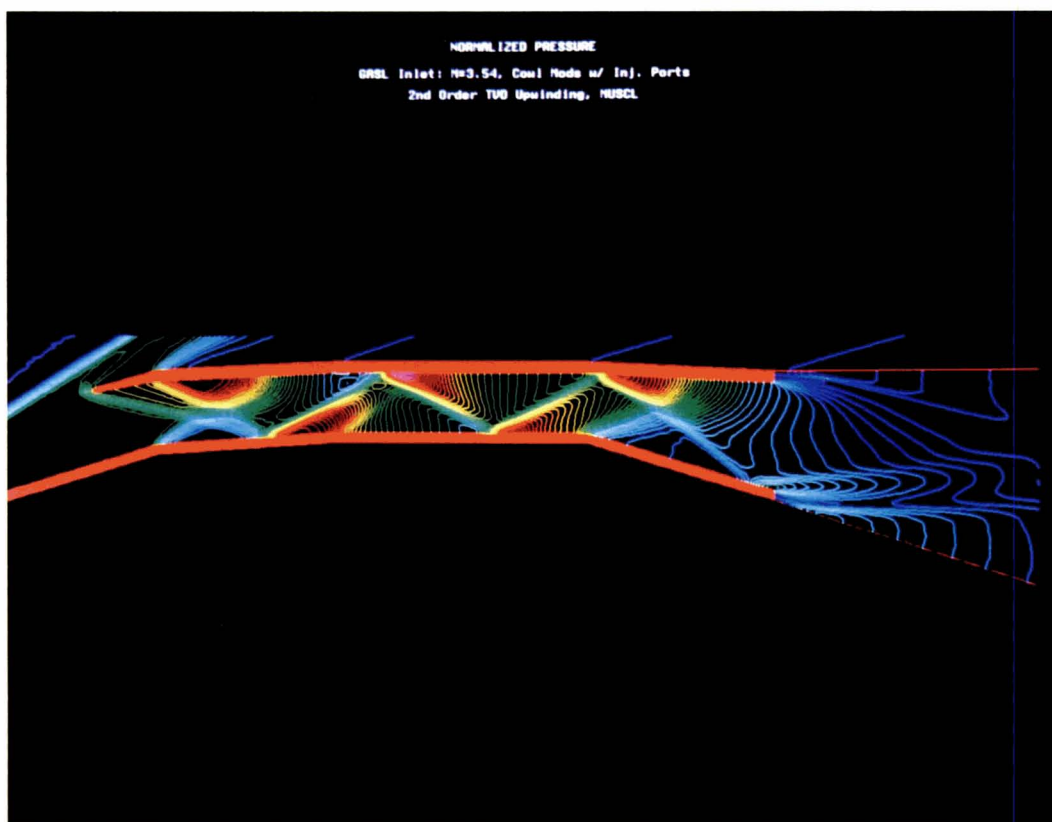
Good performance of hypersonic inlets is crucial to the viability of the hypersonic air-breathing vehicles being considered for future deployment. The ability to understand and characterize the inlet-face distortion phenomena of these configurations is limited in ground-based test facilities, which cannot yield sufficiently high resolution of the flow field and are pressed to produce flows with total enthalpies that are high enough to simulate actual flight conditions. A numerical simulation of these flow fields results in a high resolution of the underlying fluid dynamic phenomena and can account for thermodynamic effects that more closely simulate actual flight conditions.

Future Plans

Current and future state-of-the-art schemes for simulating the flows of interest with high resolution and accuracy will be developed and investigated.

Publications

1. Coirier, W. J. "High Speed Corner and Gap-Seal Computations Using an LU Scheme." AIAA Paper 89-2669, July 1989.
2. Coirier, W. J. "Efficient Real Gas Navier-Stokes Computations of High Speed Flows Using an LU Scheme." AIAA Paper 90-0391, Jan. 1990.



Normalized pressure on a GASL inlet; M = 3.54, cowl mods with inj. ports.

Transonic Navier-Stokes Solutions of Three-Dimensional Afterbody Flows

William B. Compton III, Principal Investigator

Co-investigator: Khaled Sayed Abdol-Hamid

NASA Langley Research Center/Analytical Services and Materials, Inc.

Research Objective

The goal of this project is to develop a numerical method for investigating the propulsion-integration effects of a fighter airplane at subsonic, transonic, and supersonic speeds. Intermediate steps include (1) obtaining accurate solutions for afterbodies with attached and massively separated flows, (2) including the jet exhaust in the calculations, and (3) including tails and multiple jets in the calculations.

Approach

The three-dimensional thin-layer Navier-Stokes codes CFL3D and PAB3D are used to solve for the transonic flow about a nonaxisymmetric afterbody typical of those advocated for advanced fighter airplanes. Different turbulence models are evaluated for predicting the flow in attached and massively separated regions. Comprehensive wind tunnel experiments are planned for validating the computations.

Accomplishment Description

Three algebraic turbulence models were evaluated for their ability to solve the thin-layer Navier-Stokes equations over the nonaxisymmetric nozzle at a free-stream Mach number of 1.2. The angle of attack was 0° , and the Reynolds number based on the model length was 21×10^6 . The turbulence models evaluated were the Baldwin-Lomax model, the Johnson-King model, and the Goldberg modification coupled with the Baldwin-Lomax model. All three models gave good agreement with data up to the point of shock-induced separation. The

Johnson-King and Goldberg models gave the best predictions of the separation location and the best agreement in the separated region aft of the shock. The three models predicted considerably different flow fields.

Significance

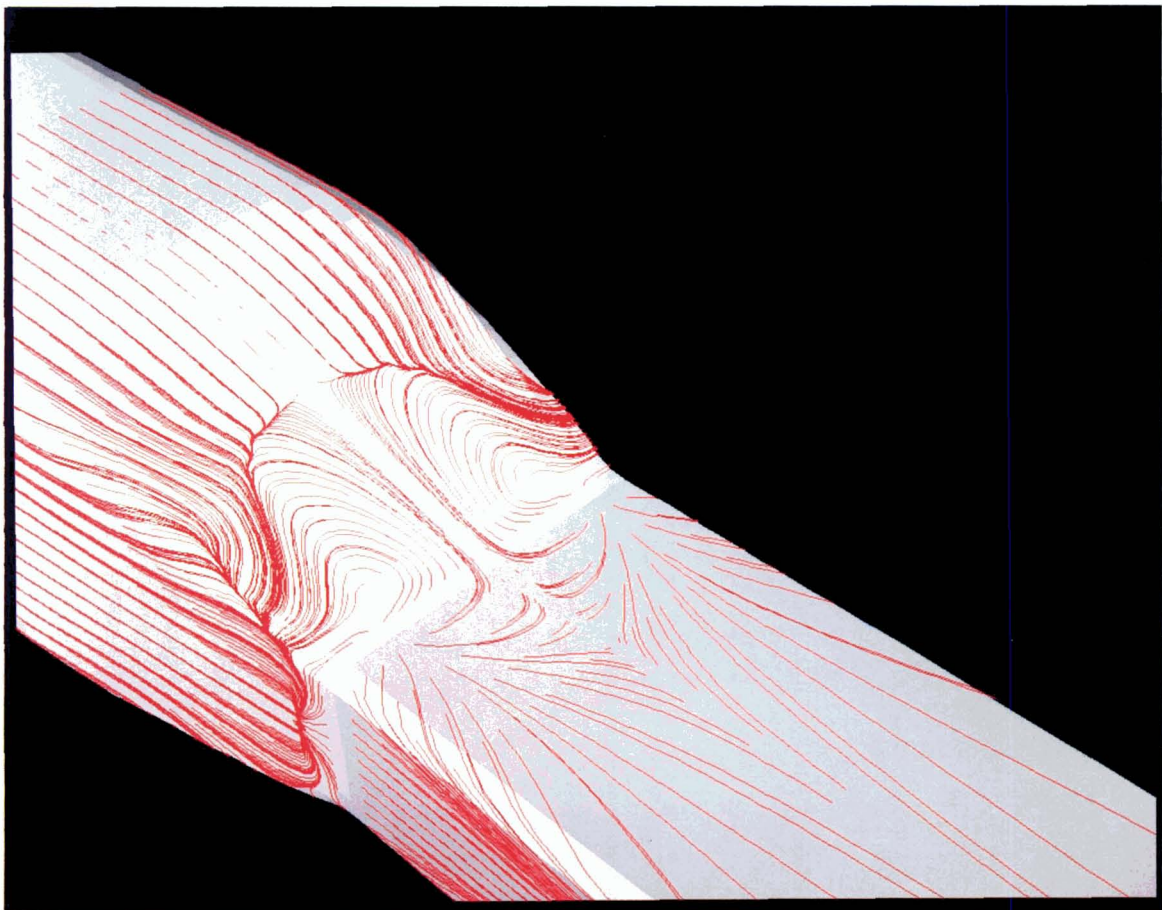
Up to one-half of the drag of a modern fighter airplane is associated with its afterbody. In the afterbody region, flows over the fuselage and empennage merge and further interact with the propulsive exhaust to create a flow field that is complex, rotational, and highly interactive. A Navier-Stokes solution technique for this problem would be very useful to designers for analyzing the afterbody flow and deriving configurations with the highest possible performance.

Future Plans

Two main efforts are proposed for the coming year. The first is to include the jet exhaust in the calculations and study its effects on the afterbody flow field. The second is to include a more advanced turbulence model and study its effect. Ultimately, the procedure will be used to study design concepts.

Publications

1. Abdol-Hamid, Khaled S., and Compton, William B. III. "Supersonic Navier-Stokes Simulations of Turbulent Afterbody Flows." AIAA Paper 89-2194, July 1989.
2. Abdol-Hamid, Khaled S., and Compton, William B. III. "Comparison of Eddy-Viscosity Turbulence Models for Supersonic Afterbody Flows." Submitted to the *AIAA J.*, Dec. 1989.



Computed oil flows for the Goldberg turbulence model; $M_\infty = 1.2$, $\alpha = 0^\circ$, $Re = 21 \times 10^6$.

Hornet 2000 Flow-Field Analysis

Raymond R. Cosner, Principal Investigator

Co-investigators: Shreekant Agrawal and Patrick J. Malloy

McDonnell Aircraft Company

Research Objective

To demonstrate the usefulness of computational fluid dynamics (CFD) in engineering studies of a current fighter aircraft, by validating computed solutions against existing wind tunnel and flight test data.

Approach

The approach is first to apply state-of-the-art CFD methods to a fighterlike aircraft, starting with a wing-alone geometry, and then to assess the impacts of grid density, interference, viscosity, and flexibility on such wing component loads as shear, bending moment, torsion, and hinge moments. Both Euler and Navier-Stokes solutions are obtained, using FLO67 and TLNS3D codes, respectively. These codes are based on explicit finite-volume central-difference formulations. The flexible-wing solutions are obtained by coupling the flow solvers with a simple NASTRAN model. Results are compared with flight data.

Accomplishment Description

All computations are performed at a free-stream Mach number of 0.90 and an angle of attack of 10° , which corresponds to a flight condition with a very high wing loading. Solutions using TLNS3D codes are obtained with a nonequilibrium half-equation Johnson-King turbulence model. Navier-Stokes results improve the predictions considerably over the inviscid results. For example, the inboard bending moment is within 8.7% of the flight data, and the torque is within 3.3%. Predicted values for shear, bending moment, and torque at the

outboard station are within 1.6%, 5.2%, and 7.6%, respectively, of the flight data. The control-surface hinge moments also show dramatic improvement over all of the inviscid results. We also find that viscosity is more important than elasticity or complete geometry in this application. Upper surface pressure contours from FLO67 and TLNS3D are shown in the accompanying figure, for both rigid and flexible wings. The CPU time for TLNS3D was 2.0 Cray-2 hours. The amount of central memory for an average job run was 20 megawords.

Significance

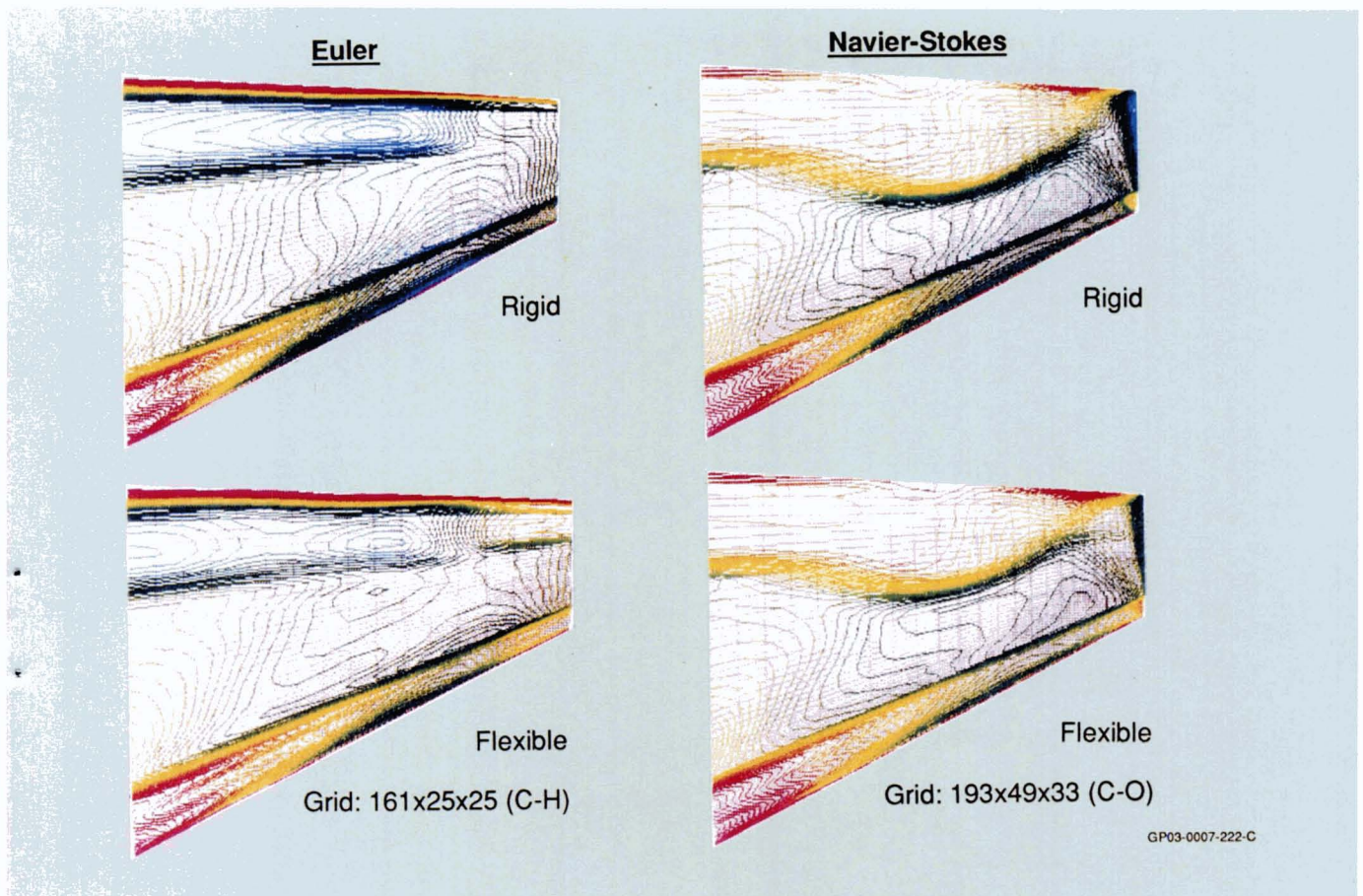
Computational fluid dynamics analysis methods of impressive power have shown high accuracy and dependability for selected classes of problems. However, the method has not been well developed for effective application of CFD to practical problems in flight-vehicle design—particularly the analysis of modern fighter-aircraft component loads at transonic and low supersonic maneuvering conditions.

Future Plans

The codes will be used to compute flow fields on more complete vehicle geometries.

Publications

Agrawal, Shreekant; Malloy, Patrick J.; and Fuglsang, Dennis F. "Design Predictions on a Fighter-Like Aircraft Wing Using Euler and Navier-Stokes Equations." Presented at the Winter Meeting of ASME, Dallas, Nov. 1990.



Pressure contours on a fighter wing; $M_\infty = 0.90$, $\alpha = 10^\circ$, $Re = 68 \times 10^6$, load factor = 7.3 g.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Dynamical Modeling of the Solar Atmosphere

Russell B. Dahlburg, Principal Investigator

Co-investigators: Jill P. Dahlburg, John T. Mariska, and J. M. Picone

Naval Research Laboratory

Research Objective

To investigate turbulent magnetohydrodynamic (MHD) processes in the solar atmosphere.

Approach

Our approach was the direct numerical simulation, using spectral methods, of the dissipative compressible and incompressible MHD equations in two- and three-dimensional geometries.

Accomplishment Description

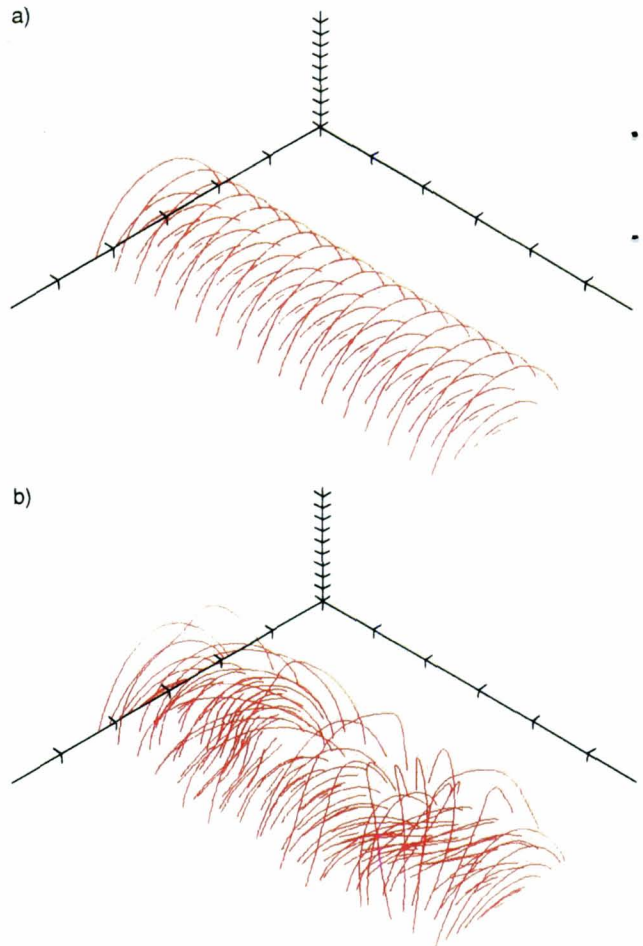
We have used our spectral method codes to investigate (1) supersonic two-dimensional MHD turbulence (typical run required 10 megawords of memory and 10 CPU hours), (2) subsonic three-dimensional compressible MHD turbulence (typical run required 23 megawords of memory and 30 CPU hours), (3) driven two-dimensional compressible turbulence (typical run required 3 megawords of memory and 20 CPU hours), and (4) three-dimensional magnetic reconnection via twisting of magnetic field lines in the solar corona (typical run required 8 megawords of memory and 20 CPU hours). The accompanying figure shows the magnetic field (a) before and (b) after twisting.

Significance

These kinds of MHD activity are thought to occur in the solar corona and the solar wind, where observations are incomplete. The numerical simulation of magnetofluids increases our understanding, especially for systems like these for which analytic work is difficult.

Future Plans

We plan to extend previous incompressible-coronal-heating calculations to the fully compressible case and to extend previous supersonic two-dimensional solar wind calculations to the fully three-dimensional case.



The magnetic field lines of the solar corona (a) before and (b) after twisting.

The Impact of Hot-Streak Migration on Turbine Heat Transfer

Roger L. Davis, Principal Investigator

Co-investigators: Daniel J. Dorney and Diane M. Rodimon

United Technologies Research Center

Research Objective

The objective of this multi-year effort is to simulate the migration of a three-dimensional combustor hot streak in the flow of an axial turbine stage and to predict the effects of the hot streak on turbine-blade surface heat transfer. A second objective is to predict blade loading distributions and the secondary flow structure in a centrifugal compressor impeller.

Approach

An extended version of the unsteady, three-dimensional Navier-Stokes code, ROTOR3, developed by Rai et al. at NASA Ames Research Center, is used to simulate three-dimensional viscous flows in axial and centrifugal turbomachinery. This approach is a third-order-accurate, upwind, approximately factored, implicit finite-difference procedure.

Accomplishment Description

A three-dimensional one-stator/one-rotor/one-hot-streak simulation with a hot-streak stagnation temperature 20% greater than the free stream was completed, and the predicted results compared with experimental data. The experimentally observed accumulation of hot fluid on the rotor pressure surface and the radial migration of the hot streak from the rotor hub to the tip were accurately captured by the numerical analysis. Two unsteady one-stator/one-rotor/one-hot-streak interaction calculations, one using specified heat-flux boundary conditions and one using inviscid boundary conditions, were initiated to help identify the roles of heat transfer and viscous effects in hot-streak migration. The calculation of three-dimensional, steady, adiabatic flow through a low-speed centrifugal compressor, with and without the effects of tip

clearance, was completed and the results compared with experimental data. The predicted blade loadings and secondary flow structures were generally in good agreement with the experimental data.

Significance

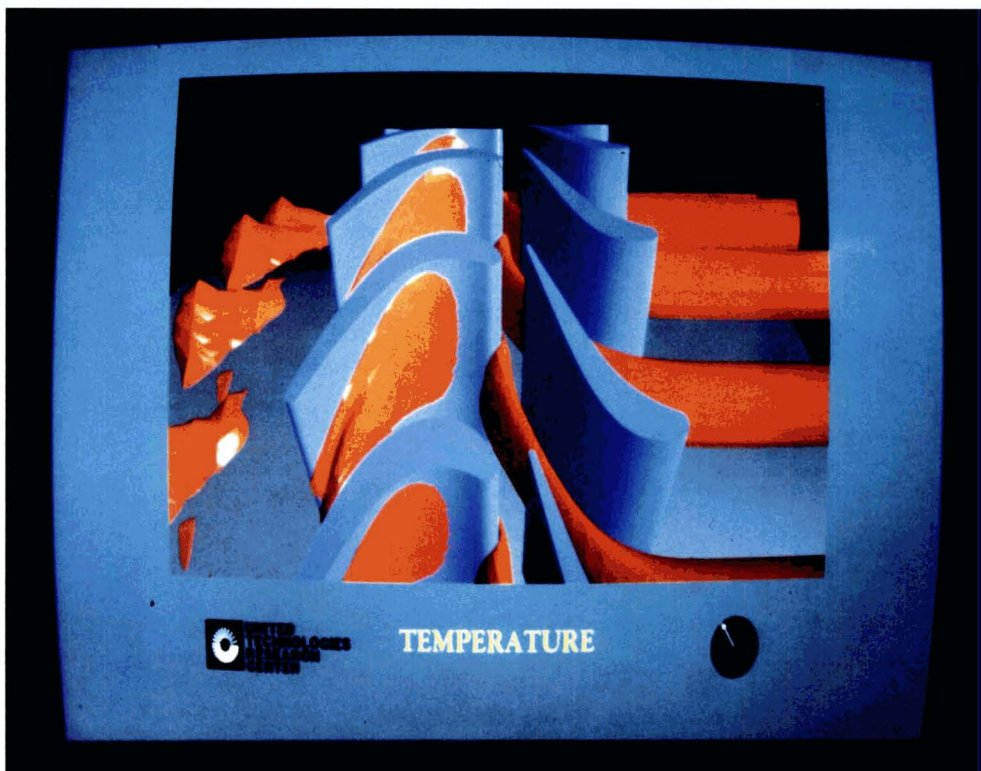
The development of a numerical procedure that can accurately predict the migration of combustor hot streaks and their effects on turbine surface heat transfer will aid turbine blade designers in determining optimum blade internal-cooling distributions and will add to the experimental data base currently available. The ability to use the same numerical analysis for centrifugal geometries has helped support the proposition of "numerical wind tunnels."

Future Plans

Upon completion of the current heat transfer and inviscid one-stator/one-rotor/one-hot-streak interaction calculations and subsequent detailed inspection of the resulting flow field, further hot-streak interaction computations will be performed to study the effects of film cooling on the turbine-blade surface heat transfer. In addition, an improved turbulence model for centrifugal geometries will be implemented, and further centrifugal compressor simulations (both steady and unsteady) will be performed. Scientific visualization will be extensively used to improve understanding of the computed flow fields in axial and centrifugal turbomachinery.

Publications

1. Dorney, Daniel J., and Davis, Roger L. "Centrifugal Compressor Impeller Aerodynamics (A Numerical Investigation)."



The impact of hot-streak migration on turbine heat transfer.

Three-Dimensional Aeroassist Flight Experiment Flow Simulations

Bill Davy, Principal Investigator

Co-investigators: Grant Palmer, Dinesh Prabhu, and Ellis Whiting

NASA Ames Research Center

Research Objective

To numerically compute the flow around the Aeroassist Flight Experiment (AFE) vehicle at one of its flight trajectory points. The results will be useful for designing the AFE base flow radiometer experiment.

Approach

The three-dimensional thermochemical nonequilibrium Navier-Stokes equations with finite-rate chemical reactions are solved using upwind-differencing fully-coupled flow codes around the AFE vehicle geometry.

Accomplishment Description

Two different codes are being used to compute flows over the AFE vehicle. The first, an explicit code, has been producing AFE flow-field solutions for over a year. The code uses a two-temperature physical model and includes 11 species and 30 chemical reactions. Comparisons with experimental and computational data have been used to validate this code. A full flow-field calculation, including the most detailed computation of the base flow to date, has been completed at the 78-km trajectory point. The explicit code's efficiency is comparable with that of implicit codes being used elsewhere. A solution over an $85 \times 23 \times 85$ grid consumes about 60 hours of Cray-2

CPU time. The second code, still in development, is a time-accurate implicit method that includes a turbulence model. When completed it will be used to study the unsteady aspects of the base region. This code also uses 11 species and uses the same two-temperature physical model as the explicit code. The accompanying figure shows vibrational temperature contours computed with the explicit code over the AFE vehicle.

Significance

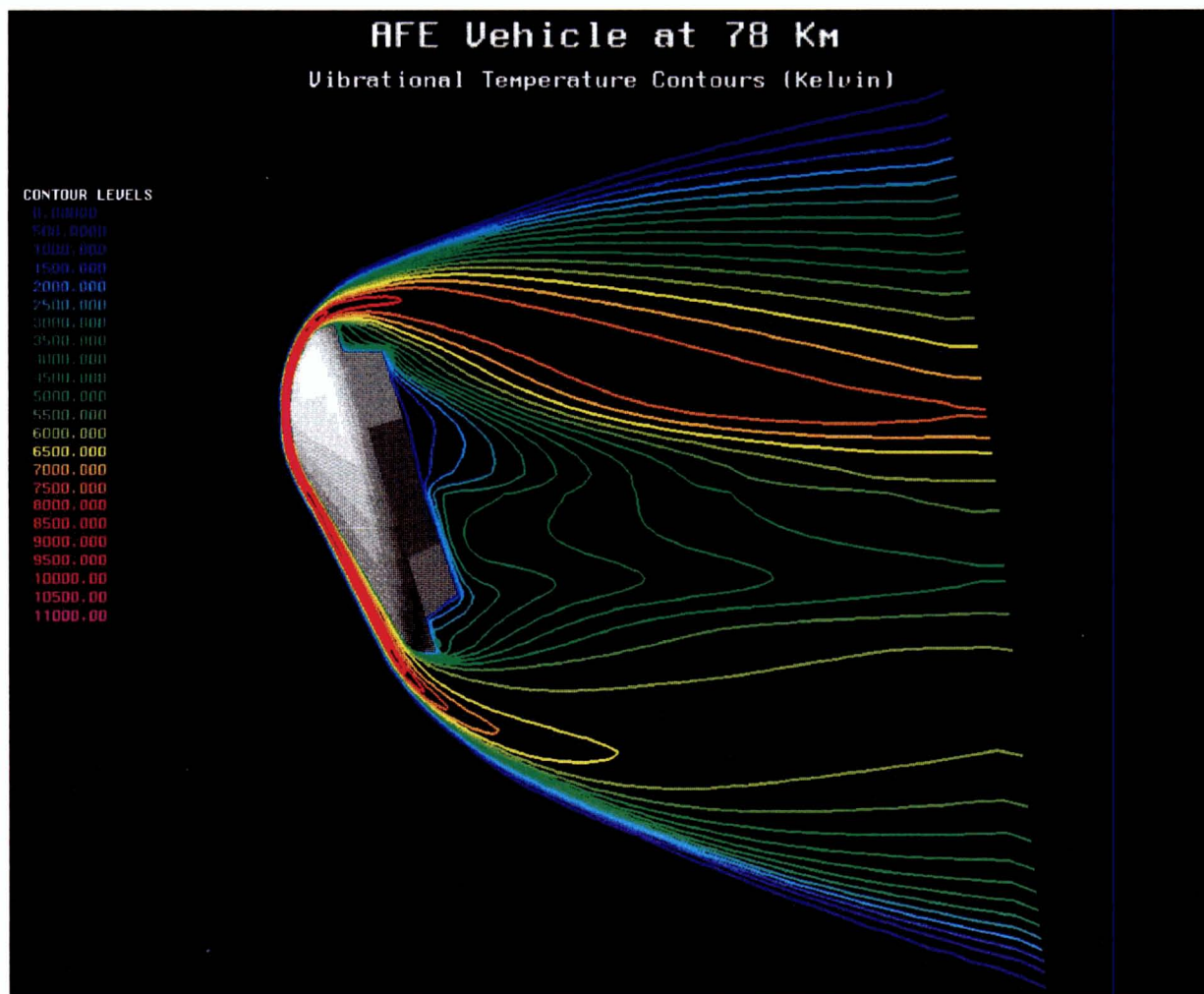
The AFE project will test the validity of using aerobraking as a fuel-saving technique for atmospheric entry and will gather data for real-gas code validation. The altitudes and velocities of the planned AFE trajectory dictate that the vehicle design and the designs of the experiments carried aboard will depend significantly on numerical simulations.

Future Plans

The codes will continue to be enhanced and used to study aspects of the AFE flow field, particularly in the base region.

Publications

Palmer, Grant. "The Development of an Explicit Thermochemical Nonequilibrium Algorithm and its Application to Compute Three-Dimensional AFE Flowfields." AIAA Paper 89-1701, June 1989.



Vibrational temperature contours (Kelvin) for the AFE vehicle at 78 km.

Integrated Fluid-Thermal-Structural Analyzer Demonstrates Heat-Transfer/Deformation Coupling

Pramote Dechaumphai, Principal Investigator
NASA Langley Research Center

Research Objective

The design of leading edges for hypersonic vehicles that experience intense stagnation-point pressures and heating rates depends on accurate prediction of the aerodynamic flow, the structural temperature response, and the structural deformations and stresses. Significant coupling occurs between the aerodynamic flow field, structural heat transfer, and structural response, creating an interdisciplinary interaction. Understanding the fluid-thermal-structural interaction is important for designing leading edges to survive such severe aerothermal environment.

Approach

An integrated fluid-thermal-structural analyzer has been developed and used to investigate the interdisciplinary interaction of an aerodynamically heated leading edge in a simulated test environment. The analyzer uses a finite-element method to solve (1) the Navier-Stokes equations for the flow solution, (2) the energy equation of the structure for the temperature response, and (3) the equilibrium equations of the structure for the structural deformation and stresses. A schematic of a proposed experimental setup of a 0.25-in.-diameter, 3-in.-long, 0.1-in.-thick leading edge is shown in the upper left figure. The leading edge is initially exposed to the undisturbed Mach 5.25 flow (position A) behind the oblique shock generated by the free-stream Mach 8 flow. After one minute, the leading edge is raised instantaneously to the predetermined position B to produce the type IV shock-shock interference pattern that results in a supersonic jet impingement normal to the leading-edge surface, causing intense stagnation-point pressure and heating rate. The fluid analysis is first performed to predict the flow behavior and aerothermal loads on the leading edge when it is at position A. An adaptive mesh-refinement technique is used in the fluid analysis to minimize the number of unknowns. The finite-element flow model and flow Mach numbers are shown in the lower left figure. The predicted aerodynamic heating rate is in excellent

agreement with the Fay and Riddell solution, as shown in the upper right figure. This heating rate causes the leading-edge temperature to increase nonuniformly, resulting in the leading edge bending upward, as shown in the centered right figure. As mentioned, the deformed leading edge is raised to the predetermined position B at one minute to produce the type IV shock-shock interference pattern. Because of the leading-edge deformation, the type IV interference pattern does not occur. The deformed shape results in the type III interference pattern, as shown in the figure. The heating-rate distribution shown in the lower right figure reflects this type III interference pattern and the high leading-edge surface temperature. The peak heating rate is caused by the shear-layer/boundary-layer interaction, and the minimum heating rate is at the nose of the leading edge, where the temperature is maximum.

Accomplishment Description

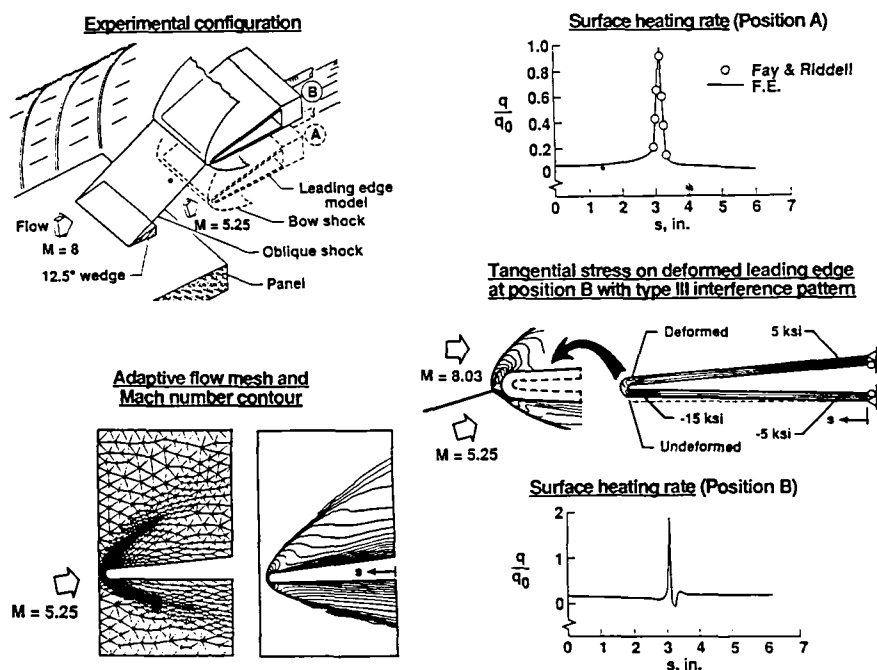
The demonstration has shown the ability of the integrated fluid-thermal-structural analysis approach to provide solutions to complex aerothermostructural behavior and the interdisciplinary interaction. The analysis computation needed about 10 Cray-2 hours and 15 megawords of memory.

Significance

The integrated fluid-thermal-structural result underlines the importance of the interdisciplinary interaction that must be considered in the design and testing of the aerodynamically heated leading edges subjected to shock-shock interference loading.

Future Plans

The integrated code will be modified to improve the structural analysis module, which includes the adaptive mesh refinement capability and implementation of unified viscoplastic theory. Also, the current computational fluid dynamics module will be applied to other structural configurations, such as sharp leading edges and compression corners.



Integrated fluid-thermal-structural analyzer demonstrates heat-transfer/deformation coupling.

Finite-Element Computation of Three-Dimensional Magnetic Fields for Design Optimization of Generators for Space Station Solar Dynamic Power

Nabeel A. O. Demerdash, Principal Investigator
Co-investigator: R. Wang
Clarkson University

Research Objective

The ultimate objective of this work is the development of a three-dimensional finite-element magnetic-field-computation code for the parameter identification and design optimization of extra high speed modified Lundell alternators (MLA). These alternators are intended for use as electric energy generators in the solar dynamic power system of NASA's Space Station Freedom.

Approach

A three-dimensional finite-element (3D-FE) computer code is being developed for the computation of the magnetic fields throughout the magnetic circuit and windings of extra high speed MLAs. This 3D-FE code is based on a novel approach that mixes magnetic vector potential and magnetic scalar potential formulations. This is in addition to the development of post processors for MLA performance computations.

Accomplishment Description

The modeling effort required the development of very complex 3D-FE grids, which discretize the entire solution region of the MLA. (See samples of armature-core, armature-winding, and rotor 3D-FE grids in the accompanying figures.) A 14.3-kVA

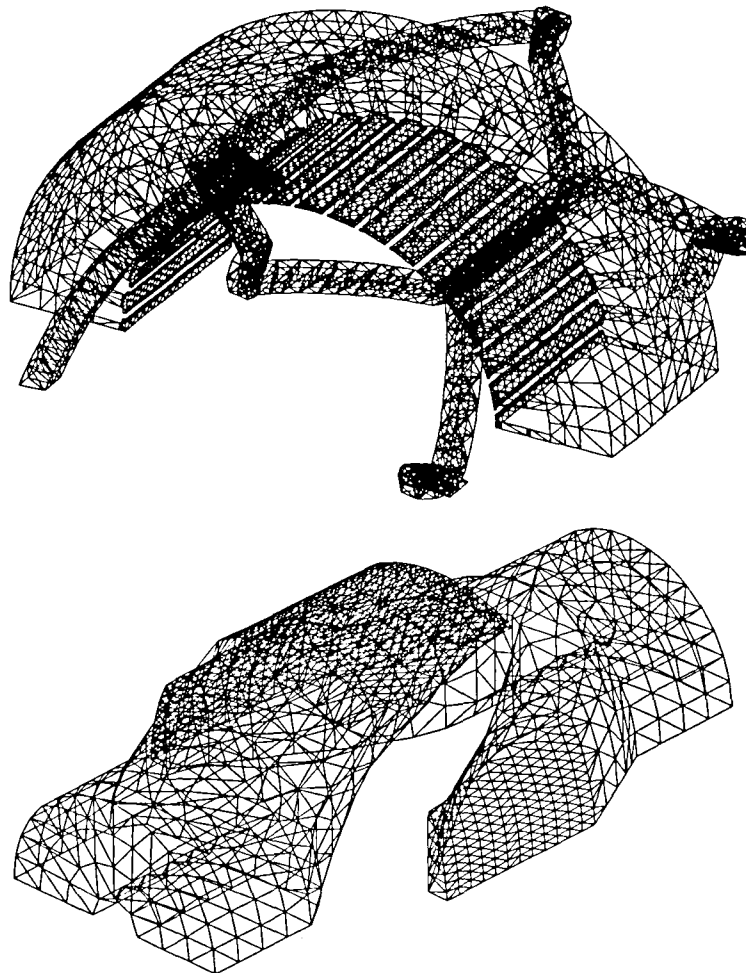
MLA is presently being analyzed using the computer code being developed in this project. Winding inductances, induced EMF waveforms, flux distributions, and open-circuit, short-circuit, and load characteristics of this machine are being obtained and analyzed using this computer code, with successful agreement between computer simulation and experimental results. The Cray-2 and Cray Y-MP were used, with 30 to 40 megawords of memory and approximately 2 hours of CPU time for a case of alternator analysis.

Significance

This computer code will be extremely useful for the design optimization and parameter identification of MLAs for space station solar dynamic power generation. This will lead to optimum generator performance and characteristics, including the volume and weight reductions that are always important in hardware that is intended for deployment in outer space.

Future Plans

Continued development and verification of this computer code is planned. This is in addition to anticipated use in actual design optimization demonstrations for space station solar dynamic MLA-type extra high speed electric generators.



Three-dimensional finite-element grids for an armature core, an armature winding, and a rotor.

High-Angle-of-Attack Inlet Analysis and Design Using Computational Fluid Dynamics Methods

Robert DeParvine, Principal Investigator
Co-investigator: Ben Smith
General Dynamics, Fort Worth Division

Research Objective

To develop the capability to apply computational fluid dynamics (CFD) to the analysis of inlet flow fields at a high-angle-of-attack flight condition.

Approach

A Navier-Stokes code is applied to a complex three-dimensional geometry at a flight condition that represents a severe condition for inlet performance. The codes are validated with detailed experimental data.

Accomplishment Description

A CFD solution of the F-16 inlet was obtained for a Mach 0.3, 40° angle-of-attack flight condition. Solutions were obtained for several inlet airflows at this flight condition. This condition causes flow separation in the inlet duct and excessively high distortion at the engine face, which are of concern for

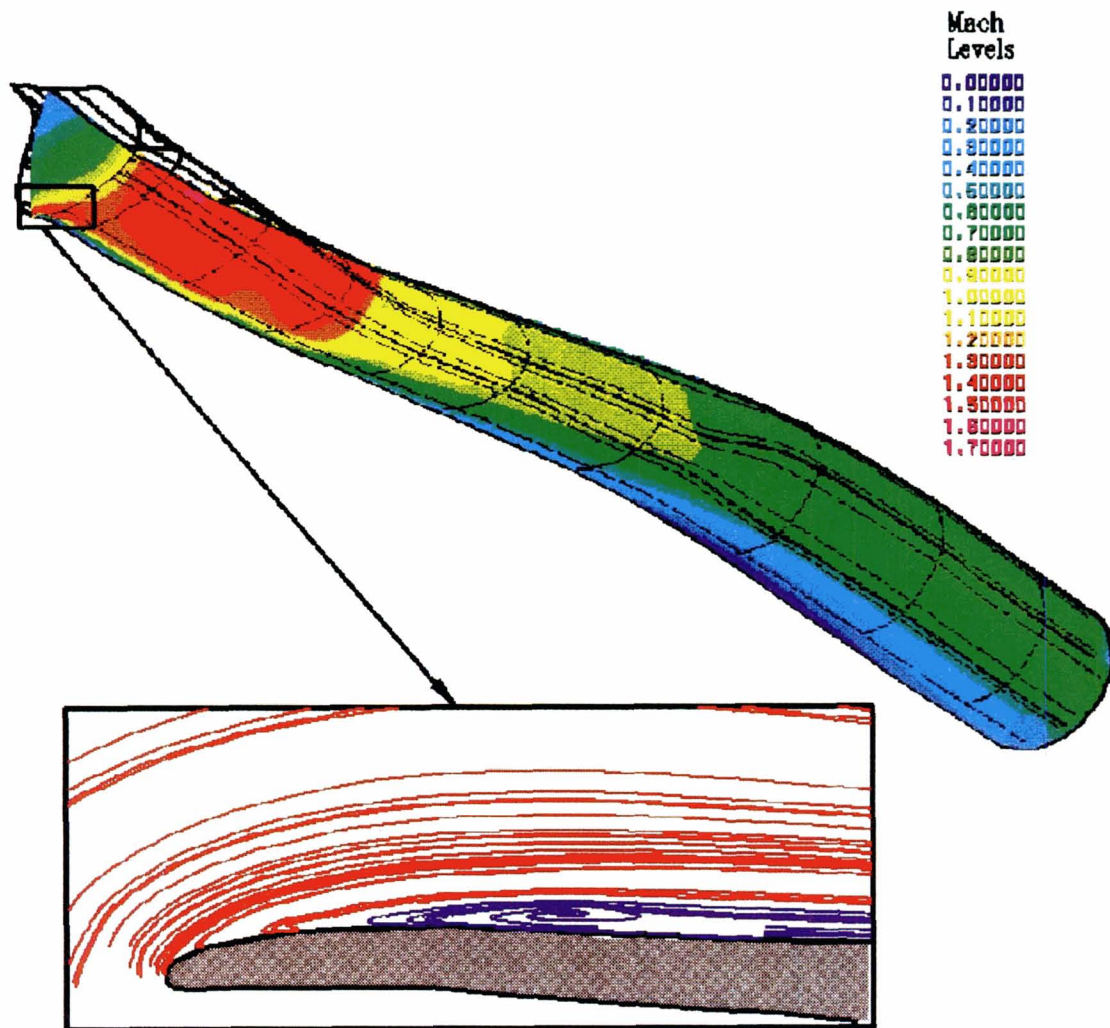
engine/inlet compatibility. Preliminary results indicate good agreement with static pressure data along the duct and with total pressure rakes located at the geometric throat.

Significance

The results of this study demonstrate a capability to predict complex, three-dimensional flow fields that have large regions of separated flow. This capability can be applied early in an inlet development program to predict inlet performance, allowing time for design refinements. Obtaining an understanding of the flow field will help to minimize risk and cost.

Publications

Howlett, Doug; McCallum, Brent; and Smith, Brian. "A CFD Study of the F-16 Inlet at High Angle-of-Attack." Presented at the AIAA Propulsion Conference, Orlando, FL, July 1990.



CFD Mach contours for the MCID-771 inlet.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Validation of a Computational Fluid Dynamics Code for High-Speed-Inlet Design

Robert DeParvine, Principal Investigator

Co-investigator: Ben Smith

General Dynamics, Fort Worth Division

Research Objective

The objective of this work is to develop the capability to apply computational fluid dynamics (CFD) to the analysis of high-Mach number inlets. The effects of an improved turbulence model are also researched.

Approach

A Navier-Stokes code is applied to inlet flows that include three-dimensional shock/boundary-layer interactions. The code is validated with detailed experimental data.

Accomplishment Description

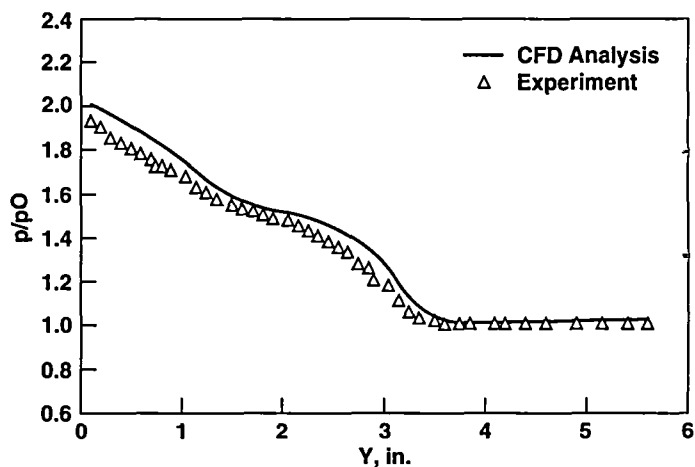
A CFD solution of the glancing shock problem was obtained for a Mach 2.94 free-stream condition. The solution shows very good agreement with wall static-pressure data and yaw-angle data. Pitot pressure data show good agreement with the CFD results. The solution shows proper flow-field features throughout the glancing shock interaction region. Strong pressure gradients within the shock/boundary-layer interaction region lead to a streamwise vortex that also is evident in the experimental results.

Significance

The results of this study demonstrate a capability to predict complex, three-dimensional, turbulent flow fields. This capability will assist in understanding complex inlet flow fields so that analysis costs and risks can be minimized.

Publications

"Air Intakes for High Speed Vehicles." To be presented at the meeting of the AGARD - FDP Working Group 13.



Surface static pressure; $x = 5.1$ in.

Advanced Solid Rocket Motor Study

Dan F. Dominik, Principal Investigator

Co-investigators: Vedat Akdag, Shmuel Ben-Shmuel, William Riba, Robert Williams, Cheng L. Chen, S. Ramakrishnan, K. Ragagopal, and S. Chakravathy

Rockwell International, Space Transportation Systems Division and Science Center

Research Objective

The objective of this work is to predict the flow field around the Space Shuttle for a modified vehicle configuration—with the advanced solid rocket motor (ASRM). The vehicle aerodynamic loads will be derived from the flow-field solution and will be used for ASRM design evaluation.

Approach

A three-dimensional Navier-Stokes code is being used to predict the multizone flow field around the Space Shuttle in ascent configurations. Both current solid rocket motor and ASRM flow fields will be calculated. Plumes will be simulated.

Accomplishment Description

An existing three-dimensional, total-variation-diminishing, full Navier-Stokes code (USA 3D) was used in the flow-field calculations of several Space Shuttle ascent configurations. Medium-resolution grids were constructed for several solid rocket booster (SRB) designs, both with and without an aft attach ring. Cases were run mainly at Mach 1.25 and -2.2° angle of attack. The results show significant effects of the attach ring on the lower-wing surface pressures. Each flow field contained approximately 300,000 grid points, and took about 35 combined Cray Y-MP and Cray-2 hours and 12 megawords of memory.

Significance

The proposed ASRM vehicle will significantly upgrade the Space Shuttle performance and increase flight safety margins. The numerical simulations will be used to evaluate design configurations and help in the final selection of design candidates for wind tunnel testing.

Future Plans

The Space Shuttle model is being expanded to include all geometric components affecting the flow field. Also, the grids are being refined in order to increase geometric and computational accuracies. The number of grid points are expected to increase to over 1 million for zero-sideslip-angle cases and will be double that number for nonzero-sideslip-angle solutions. Solutions will be obtained for several Mach numbers, angles of attack and sideslip, and with and without plume simulation.

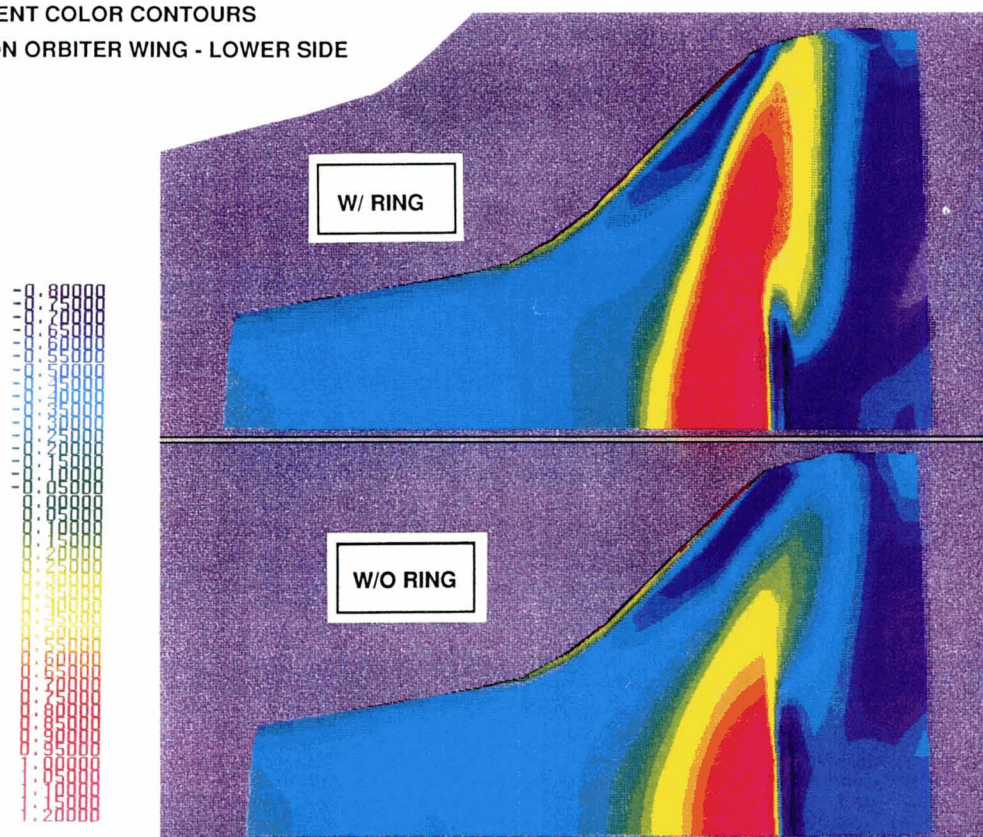
Publications

1. Ragagopal, K. et al. "Versatile Multi Zonal Gridding Technique for Complex Geometries." AIAA Paper 90-0011, Jan. 1990.
2. Chen, C. L. et al. "Multi Zonal Navier-Stokes Solutions for the Multi Body Space Shuttle Configuration." AIAA Paper 90-0434, Jan. 1990.

RI - FNS Solution at Mach = 1.25 , $\alpha = -2.2^\circ$

PRESSURE COEFFICIENT COLOR CONTOURS

SRB RING EFFECTS ON ORBITER WING - LOWER SIDE



ORIGINAL PAGE
COLOR PHOTOGRAPH

Pressure coefficient color contours showing SRB ring effects on the lower surface of the orbiter wing; RI-FNS solution at $M = 1.25$, $\alpha = 2.2^\circ$.

Simulation of Supersonic Chemically Reacting Flow Fields

J. Philip Drummond, Principal Investigator

Co-investigators: Mark H. Carpenter, Peyman Givi, Johnny R. Narayan, David W. Riggins, Balu Sekar, and Jeffrey A. White
NASA Langley Research Center

Research Objective

To study phenomena that control high-speed fuel-air mixing and combustion, and to use information gained from these studies to improve mixing and combustion efficiency in high-speed propulsion devices.

Approach

The combustion processes occurring in a scramjet combustor are described by using an accurate numerical algorithm to solve the equations governing the mixing and chemical reactions of a multicomponent mixture in combination with a physically realistic chemical kinetics model.

Accomplishment Description

A computer program was developed during the previous period that solved the equations governing a multicomponent mixing and reacting flow. The code was validated against experiments involving both nonreacting and reacting flow fields, and then applied in studies of combustor high-speed engine flows. Recent research has been directed toward improving the efficiency of fuel-air mixing and reaction in a scramjet. Mixing is significantly reduced in this engine as the combustor Mach number increases with flight Mach number, and mixing enhancement is required to achieve a sufficient degree of combustion efficiency. Because of this difficulty, alternate combustor fuel-injector configurations were designed and studied, to evaluate their potential for producing an improved degree of mixing and reaction. One such configuration is shown in the figure. It consists of two swept-ramp fuel injectors that introduce gaseous hydrogen fuel from orifices in their bases. The swept-ramp sidewalls induce streamwise vorticity into the inlet air passing the ramps, and this vorticity enhances fuel-air mixing downstream of the injectors. The resulting hydrogen-fuel mass fraction distribution, keyed to the color bar, is also shown in the figure. Without enhancement,

the fuel jet would spread only slightly over the distance studied. With enhancement, however, the fuel jets are captured by the vortices and spread well across the channel downstream of the ramps. This dramatic improvement in fuel-air mixing results in a significant improvement in combustion efficiency. A typical run for this problem on the Cray Y-MP required approximately 8 hours, and 40 megawords of memory.

Significance

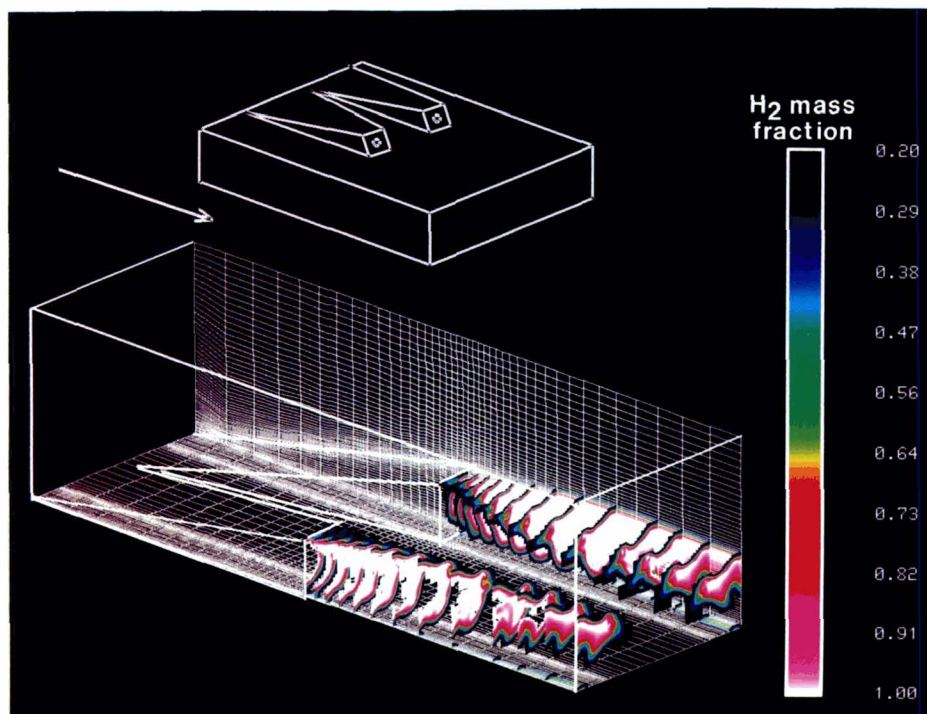
Numerical simulations provide improved insight into the mechanisms controlling high-speed mixing and combustion in supersonic reacting flows. Using knowledge gained from these basic simulations, techniques can be developed for enhancing mixing and reaction in supersonic combustors, thus improving the overall level of combustion efficiency in these devices.

Future Plans

Work is now under way to optimize the degree of enhancement that can be achieved from the fuel-injector configuration considered in this study. Other injector configurations that utilize shock excitation and vortex bursting for enhancement will also be examined.

Publications

1. Drummond, J. P.; Carpenter, M. H.; Riggins, D. W.; and Adams, M. S. "Mixing Enhancement in a Supersonic Combustor." AIAA Paper 89-2794, July 1989.
2. Carpenter, M. H. "Three-Dimensional Computations of Cross-Flow Injection and Combustion in a Supersonic Flow." AIAA Paper 89-1870, June 1989.
3. Riggins, D. W.; Mekkes, G. L.; McClinton, C. R.; and Drummond, J. P. "A Numerical Study of Mixing Enhancement in a Supersonic Combustor." AIAA Paper 90-0203, Jan. 1990.



An alternate combustor fuel-injector configuration.

Acceleration of Iterative Algorithms for Euler and Navier-Stokes Equations

George S. Dulikravich, Principal Investigator
Co-investigator: Seungsoo Lee
The Pennsylvania State University

Research Objective

To develop a reliable algorithm that gives a faster convergence rate than conventional iterative algorithms, and to investigate its dependence on grid and flow conditions.

Approach

The distributed minimal residual (DMR) method has been developed and successfully applied to the system of Euler equations and the system of incompressible Navier-Stokes equations. The DMR method belongs to a general class of Krylov subspace methods. However, it has two aspects that are different than the Krylov subspace methods. First, a different sequence of acceleration factors is used for each equation of the system; and second, only two to five steps are needed to achieve good convergence rates.

Accomplishment Description

The convergence of the conventional explicit and implicit algorithms (Runge-Kutta time-stepping method and Euler implicit method) have been accelerated. It has been shown that the CPU time to achieve steady state for two-dimensional problems has been reduced to 20-70% of the time needed by the nonaccelerated schemes. The DMR method has been extensively tested on the sensitivity to grid and flow conditions, including grid clustering, grid nonorthogonality, number of grid points, Mach number, and Reynolds number. The sensitivity of the DMR method to CFL number has been tested and the comparison with the implicit residual smoothing method has been performed as well. The DMR method has been proven to be less sensitive to grid and flow conditions than the existing methods.

Significance

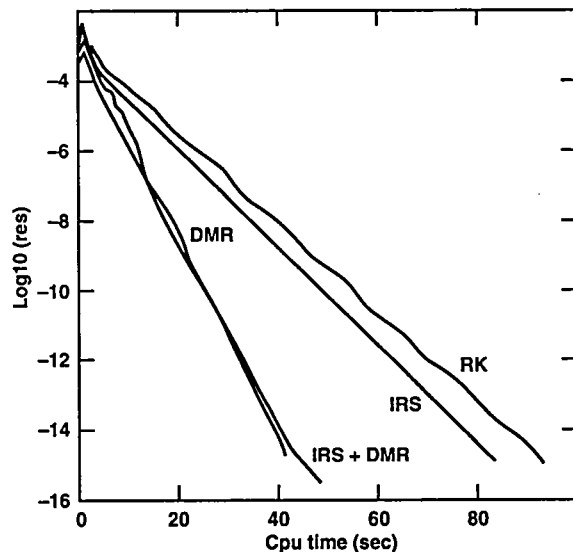
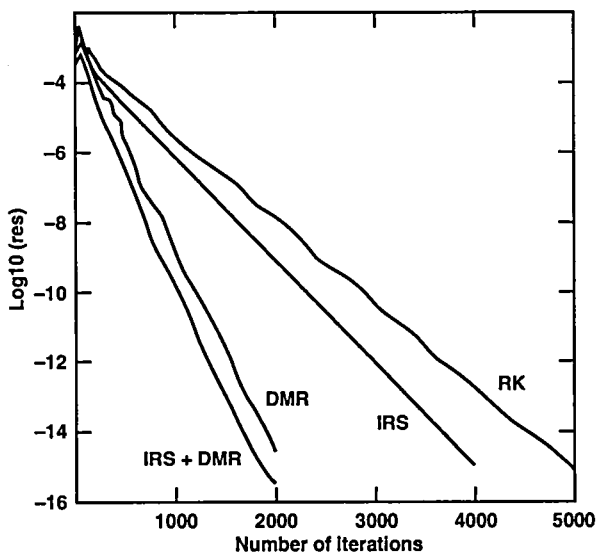
As a tool of computational fluid dynamics, the importance of computational algorithms cannot be emphasized too much. The reliability of a computational algorithm depends on its convergence rate and robustness. The performance test of a newly developed scheme is of paramount importance.

Future Plans

A three-dimensional explicit code has been developed and verified against experimental data and published computational results. The DMR method is being incorporated into this three-dimensional code. The DMR method will be evaluated for its ability to enhance the convergence of three-dimensional problems, through a number of tests. Especially, the DMR method will be examined for its ability to accelerate the convergence of turbulent flow computations. A three-dimensional implicit code will be written and used to evaluate the DMR method.

Publications

1. Lee, Seungsoo, and Dulikravich, George S. "A Fast Iterative Algorithm for Incompressible Navier-Stokes Equations." *Proceedings of the 10th Brazilian Congress of Mechanical Engineering*. Rio de Janeiro, Brazil, Dec. 1989.
2. Lee, Seungsoo, and Dulikravich, George S. "Distributed Minimal Residual (DMR) Method for Accelerations of Iterative Algorithms." *Proceedings of the Computational Fluid Dynamics Symposium on Aeropropulsion*. NASA CP-10045, 1990.
3. Lee, Seungsoo, and Dulikravich, George S. "Performance Analysis of the DMR method for Accelerations of Iterative Algorithms." Presented at the 27th Aerospace Sciences Meeting, Reno, NV, Jan. 1991.



Convergence histories of the Runge-Kutta method for Hiemenz flow; Re = 400.

The Interaction of Particles with Homogeneous Turbulence

John K. Eaton, Principal Investigator
Co-investigator: Kyle D. Squires
Stanford University

Research Objective

The objective of this work is to gain an increased physical understanding of the interaction of small, solid particles with gas-phase turbulence. This understanding will lead to improved turbulence models for engineering computations of particle-laden turbulent flows.

Approach

The interaction of particles with stationary isotropic turbulence is studied using direct numerical simulation of the three-dimensional, time-dependent, incompressible Navier-Stokes equations. A vectorized interpolation algorithm has been developed to facilitate the simultaneous tracking of as many as one million particles through the simulated flow fields.

Accomplishment Description

Results from the simulations showed significant effects of turbulent motions on particle concentration fields. Particles were shown to collect in regions of low vorticity and high strain rate. This bias in particle motion was quantified using various correlations of the particle number density with turbulence quantities. Instantaneous values of the particle number density were shown to be as much as 30 to 40 times the mean value because of particle accumulation in these regions. The effect

of particle loading on turbulent flow fields was also investigated in the simulations. It was found that the decrease in turbulence kinetic energy was relatively insensitive to particle inertia, being dependent only on the value of the mass loading. A computation of turbulence modification using 64^3 points for the hydrodynamic calculation and 10^6 particles required approximately 20 Cray-2 hours and approximately 35 megawords of memory.

Significance

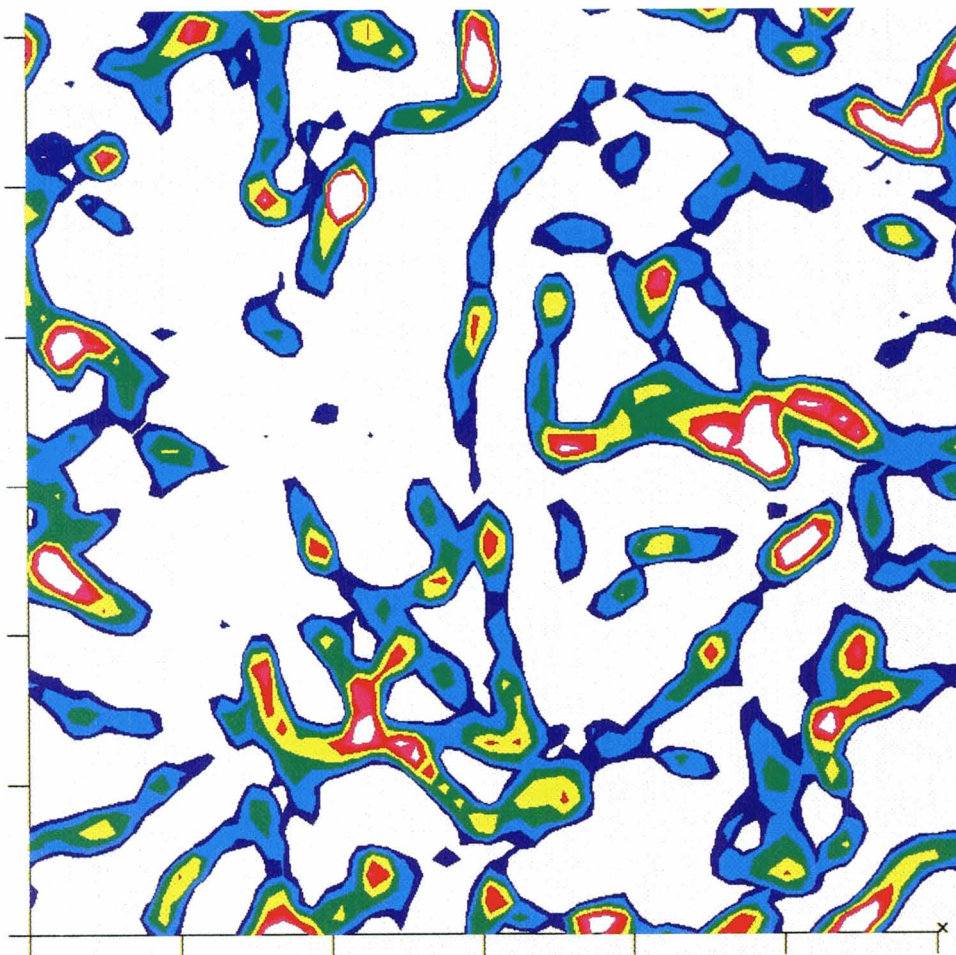
The ability to use direct numerical simulation of turbulence to compute turbulence modification by particles has been demonstrated. The data base generated from these computations may be used as a basis for further developments in turbulence modeling for particle-laden turbulent flows.

Future Plans

Future plans include the examination of current turbulence models of particle transport in turbulent flow fields as well as the examination of turbulence modification by particles.

Publications

Squires, Kyle D., and Eaton, John K. "Particle Response and Turbulence Modification in Isotropic Turbulence." To be published in *Physics of Fluids*.



Contours of the particle number-density field from one plane of a direct numerical simulation of the three-dimensional, incompressible Navier-Stokes equations. The contour levels are from the mean value to four times the mean number density for a dimensionless particle time constant of 0.15.

Hypersonic Chemically Reacting Flow over Blended Wing-Body Vehicles

Thomas A. Edwards, Principal Investigator
Co-investigators: Jolen Flores, Uwe Jettmar, and Greg Molvik
NASA Ames Research Center

Research Objective

The goal of this research is to establish a computational fluid dynamics (CFD) capability to solve external hypersonic flows about candidate vehicle configurations, such as the National Aero-Space Plane (NASP). This requires adding physical models for transition and for the chemical reactions that take place in air when it is heated by the strong shock waves associated with hypersonic flight. Once these models are implemented, the code must be validated by comparing the calculations with experimental data.

Approach

The compressible Navier-Stokes (CNS) code is being developed in support of the NASP program. This code was enhanced by incorporating a simple transition model, a model for air in chemical equilibrium, and a set of equations for air in chemical nonequilibrium. These modifications were validated by comparing calculated results with experimental data.

Accomplishment Description

The CNS code now includes the elements needed to simulate the external hypersonic flight environment about a vehicle such as the NASP. All features and options of the code have been exercised and validated with experimental data. This is especially important for hypersonic vehicle design because the acquisition of data via experimental techniques is limited and costly. A typical CFD calculation of hypersonic, chemi-

cally reacting flow past a blunt body requires about 10 hours of Cray-2 CPU time and 4 megawords of main memory.

Significance

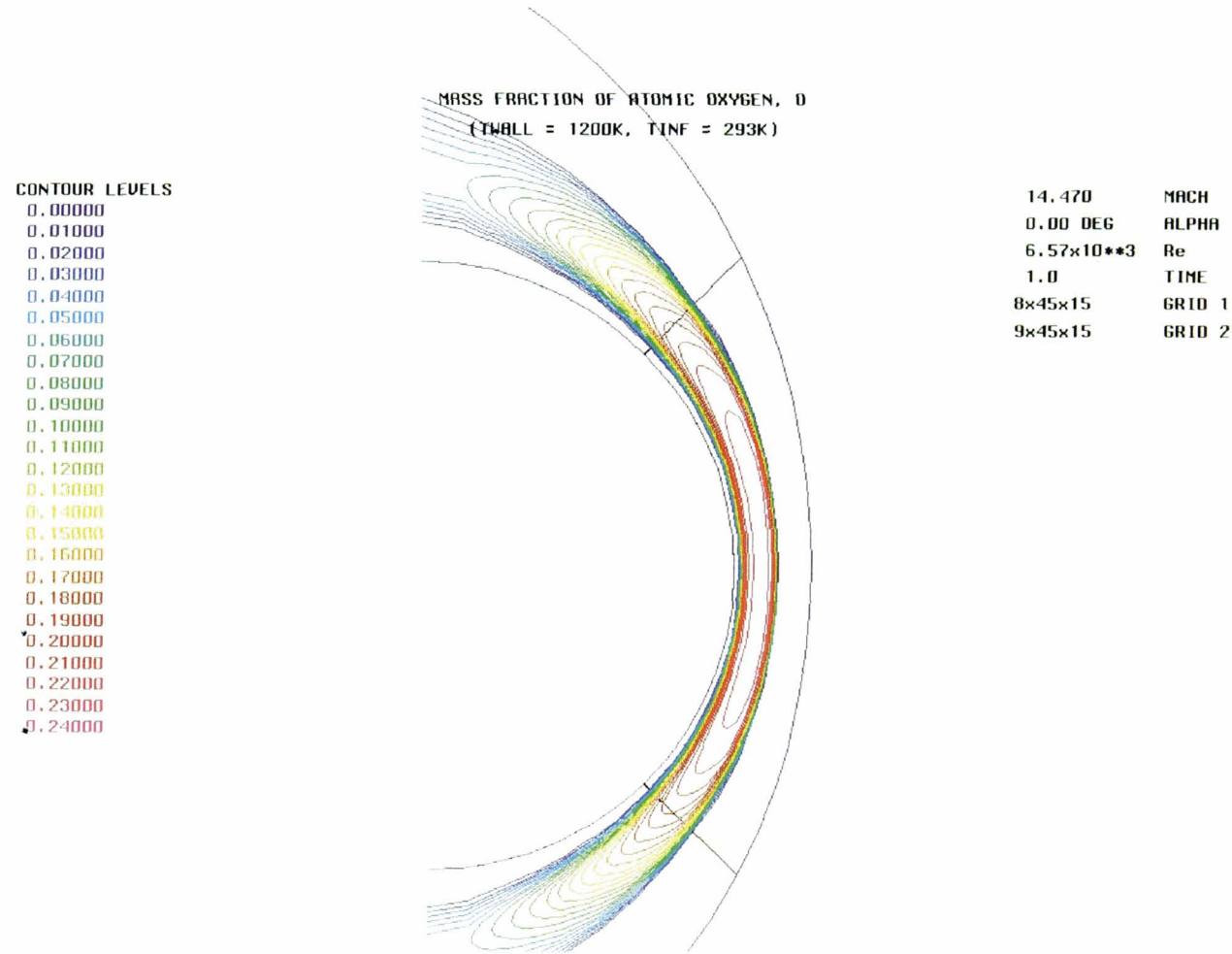
The CNS code is among the first CFD codes with the ability to address chemically reacting, hypersonic flows past geometrically complex shapes. This capability will contribute to the design of the NASP.

Future Plans

The CNS code will continue to be validated as experimental data becomes available. A higher order turbulence model will be incorporated, and a solution-adaptive grid method will be applied. Vectorization and convergence-acceleration techniques will be explored.

Publications

1. Flores, J.; Edwards, T. A.; Ryan, J. S.; and Kaul, U. "CNS Code Enhancements (U)." Paper 15, 6th NASP Symposium, Apr. 1989.
2. Edwards, T. A., and Flores, J. "Toward a CFD Nose-to-Tail Capability: Hypersonic Unsteady Navier-Stokes Code Validation." *J. Spacecraft Rockets* 27, no. 2 (Mar.-Apr. 1990): 123-130. (Also: AIAA Invited 1-hour paper 89-1672, June 1989.)
3. Edwards, T. A. "Numerical Simulation of Hypersonic Reentry Flow Fields." Presented at the Workshop on Hypersonic Flows for Reentry Problems, Antibes, France, Jan. 1990.



Hypersonic chemically reacting flow past a blunt body.

Numerical Simulation of Hydrogen-Air Combustion in Hypersonic-Vehicle Propulsion Systems

Thomas A. Edwards, Principal Investigator
Co-investigators: Jolen Flores and Uwe Jettmar
NASA Ames Research Center

Research Objective

To develop and validate a computational fluid dynamics (CFD) capability to solve the flow through the integrated propulsion system of air-breathing hypersonic-flight vehicles, and to incorporate the boundary conditions and physical models necessary to obtain accurate solutions to flows involving hydrogen-air combustion, including the recombination-dominated external flow on the afterbody.

Approach

First, the compressible Navier-Stokes (CNS) code is extended to address forebody/inlet flow fields. Because the vehicle propulsion system is highly integrated, the external and internal flow fields cannot be analyzed separately. Next, a hydrogen-air combustion code is obtained and coupled to address the internal portion of the flow. Finally, the CNS code is extended to model the recombination-dominated flow on the afterbody.

Accomplishment Description

The CNS code demonstrated its ability to solve mixed external/internal flow fields on two configurations, one of which was a generic forebody/inlet shape characteristic of the National Aero-Space Plane (NASP). Correlation with the experimental data was very good. The afterbody chemistry modeling has been completed and is in the process of being incorporated into the CNS code. The internal combustion codes available for inclusion are being evaluated. Power-off calculations about representative configurations require about

50 hours of Cray-2 CPU time, 4 megawords of main memory, and 20 megawords of secondary memory. It is expected that a fully modeled power-on simulation will require about four times this amount of CPU time and twice the memory.

Significance

The CNS code is among the first CFD codes to demonstrate the ability to solve the entire flow about candidate NASP configurations, including the forebody, inlet, nozzle, and afterbody in a fully-coupled solution. This is the kind of analysis necessary to predict performance of proposed NASP designs. This kind of information cannot be obtained in such detail by any other means.

Future Plans

A demonstration calculation will be performed for a generic NASP configuration in powered, hypersonic flight. Validation cases will be identified and calculations performed for them, and the CNS code will be applied to support the NASP program.

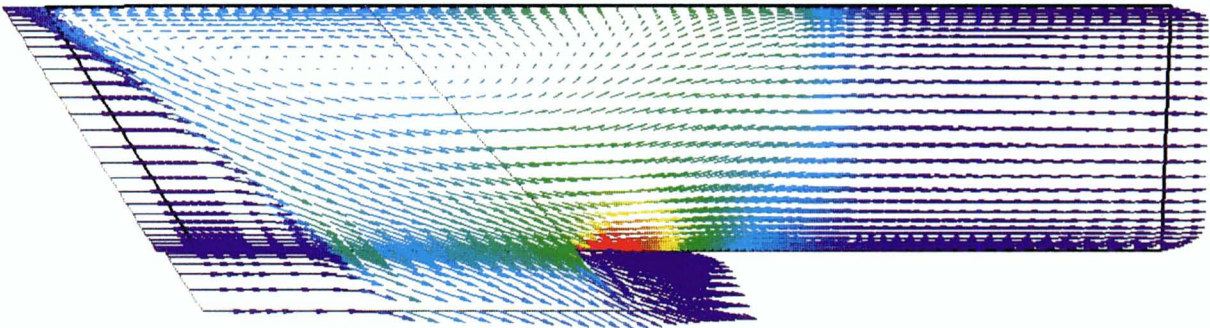
Publications

1. Bennett, B. C., and Edwards, T. A. "Calculation of Hypersonic Forebody/Inlet Flow Fields (U)." Paper 48, 7th National Aero-Space Plane Symposium, Oct. 1989.
2. Bennett, B. C., and Edwards, T. A. "Numerical Solution of TTD Forebody/Inlet Flow Fields (U)." Paper 47, 8th National Aero-Space Plane Symposium, Mar. 1990.

FIVE ZONE VELOCITY VECTORS(U)

CONTOUR LEVELS
0.00000
25.00000
50.00000
75.00000
100.0000
125.0000
150.0000
175.0000
200.0000
225.0000
250.0000
275.0000
300.0000
325.0000
350.0000
375.0000
400.0000
425.0000
450.0000
475.0000
500.0000
525.0000
550.0000
575.0000
600.0000
625.0000
650.0000
675.0000
700.0000
725.0000
750.0000
775.0000

1.667 GAMMA
18.100 MACH
0.00 DEG ALPHA
3.86x10**5 Re
9.65x10**2 TIME
8x61x36 GRID 1
16x61x22 GRID 2
52x61x22 GRID 3
21x13x22 GRID 4
12x23x22 GRID 5



Five-zone velocity vectors (U).

ORIGINAL PAGE
COLOR PHOTOGRAPH

Advanced CFD Codes for Rotary-Wing Airloads and Performance Prediction

T. Alan Egolf, Principal Investigator
Co-investigator: Brian E. Wake
United Technologies Research Center

Research Objective

To refine and enhance advanced helicopter and high-speed-propeller computational fluid dynamics (CFD) codes and wake-modeling codes in order to include more rigorous representations of the rotor wake and provide viscous drag prediction capability for improved airload and performance prediction.

Approach

Rotor-wake influence is being incorporated into rotary-wing CFD codes using velocity-embedding techniques and transpiration boundary conditions based on prescribed wake geometries from either analytical, empirical, or numerical predictions. Improved wake-geometry prediction capability is being developed based on free-wake vortex-lattice methods.

Accomplishment Description

Activity during this period of the multi-year effort focused on the application of the United Technologies Research Center's viscous code NSR3D to complex geometries. The applications were directed toward initial investigation of the ability of the code to predict the flow-field behavior associated with vortex structures generated by such geometries as a helicopter blade with a BERP-like tip planform, a high-speed propeller, and a fixed-wing configuration. Tip and leading-edge vortex structures were predicted. Approximately 15 to 25 hours of Cray-2 processor time and about 30 megawords of memory were required per case. In addition, the computational performance of the NSR3D and wake prediction codes used in this

effort is being compared with their performance on other advanced computer architectures.

Significance

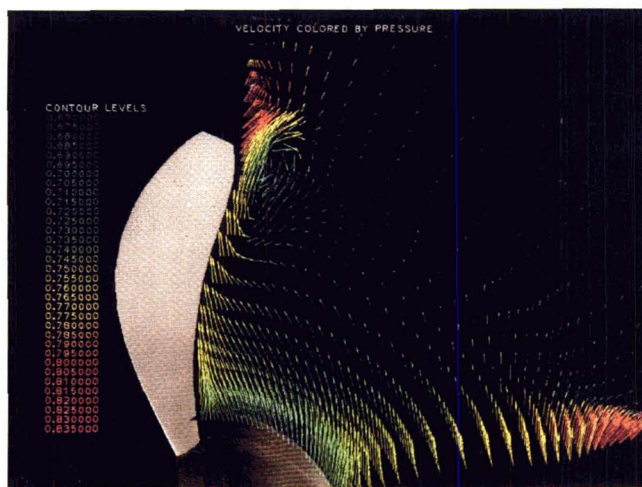
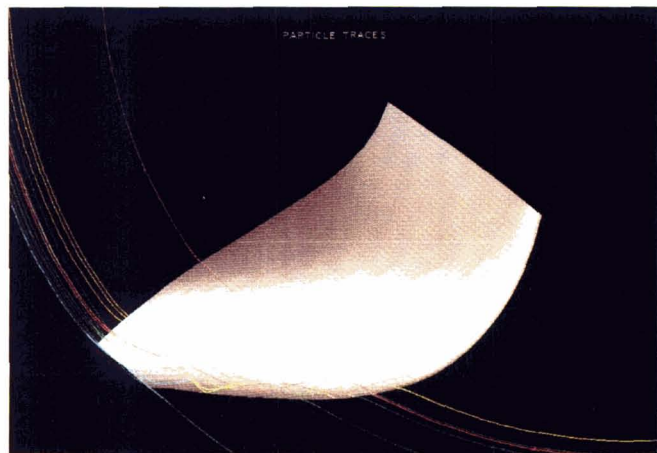
Application of the viscous code NSR3D as part of the development and verification effort has demonstrated the prediction of fluid flow phenomena associated with real-world geometries.

Future Plans

Refinements to the solver are planned along with the incorporation of the hover-wake influence into the viscous code NSR3D using a refined hover free-wake model. Application of the NSR3D code to an advanced blade tip for acoustic reduction purposes is also planned under a NASA contract.

Publications

1. Egolf, T. A. "Helicopter Free-Wake Analysis on a Massively Parallel Computer." *Proceedings of the International Symposium on Boundary Element Methods 89*, Oct. 1989.
2. Egolf, T. A. "Connection Machine Utilization and Experience at the United Technologies Research Center." *Proceedings of the Fourth SIAM Conference on Parallel Processing for Scientific Computing*, Chicago, Dec. 1989.
3. Wake, B. E., and Egolf, T. A. "Implementation of a Rotary-Wing Three-Dimensional Navier-Stokes Solver on a Massively Parallel Computer." AIAA Paper 89-1939, June 1989.
4. Wake, B. E., and Egolf, T. A. "Application of a Rotary-Wing Three-Dimensional Navier-Stokes Solver on a Massively Parallel Computer." AIAA Paper 90-0334, Jan. 1990.



(Left) A leading-edge vortex structure for a high-speed propeller, predicted by NSR3D and demonstrated by particle traces.
(Right) The velocity field in a plane downstream of the trailing edge.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Direct Simulation of Turbulent Reacting Flows

S. E. Elghobashi, Principal Investigator
Co-investigator: K. K. Nomura
University of California, Irvine

Research Objective

To study the physical processes of turbulent mixing in a non-premixed flame with and without the effects of heat release.

Approach

Direct numerical simulation of the full, three-dimensional, time-dependent Navier-Stokes, continuity, and scalar conservation equations are performed. The general equations are valid for both incompressible and compressible low-Mach-number flows. Initially, nonpremixed reactants in both isotropic and homogeneous shear flows are considered.

Accomplishment Description

Simulations were performed for an isothermal (negligible heat release) nonpremixed reaction in both isotropic and homogeneous shear flows. These simulations provided detailed statistics on vorticity, strain rate, and scalar dissipation rate fields; the results described their roles and interactions in scalar mixing. Simulations using a resolution of 96^3 grid points require approximately 2 to 3 hours on the Cray-2 and 15 megawords of memory.

Significance

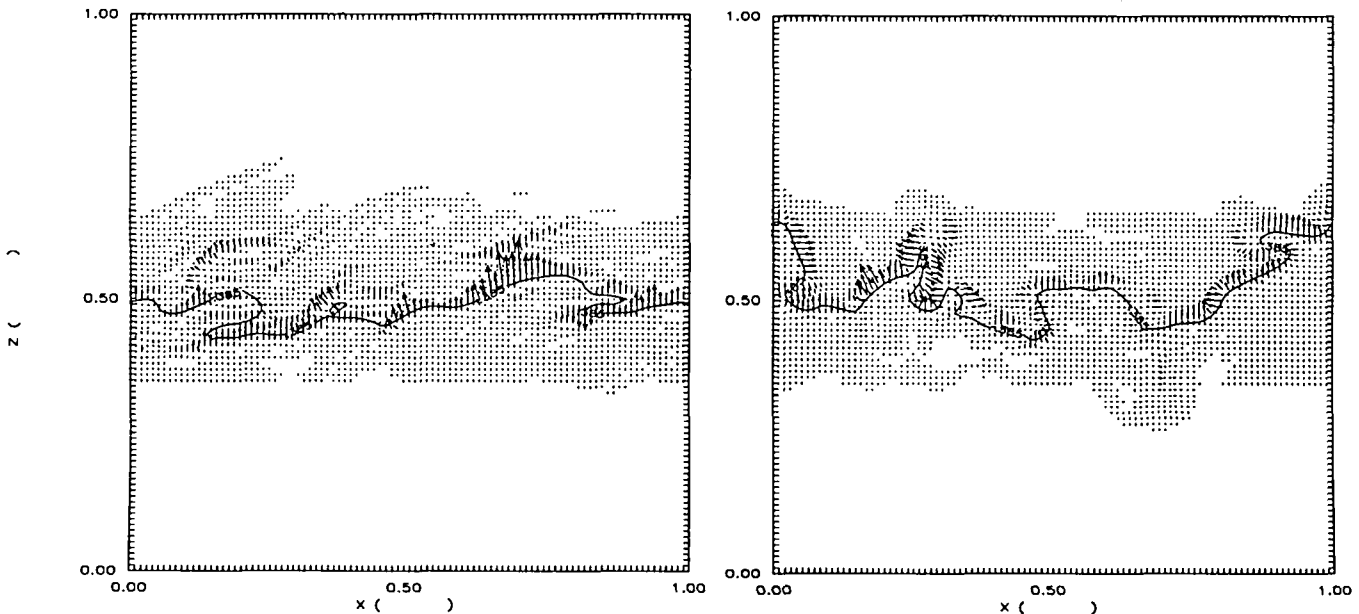
The interaction of heat release and turbulence is one of the least understood phenomena in chemically reacting flows. Results obtained from direct numerical simulations that accommodate the effects of variable density due to heat release will provide a more accurate description of the flame structure and turbulent statistics and will contribute to our understanding of these flows.

Future Plans

Work is under way to extend the simulations to reactions with heat release.

Publications

1. Nomura, K. K., and Elghobashi, S. E. "Direct Simulation of an Isothermal Nonpremixed Flame in Homogeneous Turbulent Shear Flow." AIAA Paper 90-0148, 1990.
2. Elghobashi, S. E., and Nomura, K. K. "Direct Simulation of a Passive Diffusion Flame in Sheared and Unsheared Homogeneous Turbulence." In *Turbulent Shear Flows 7*, ed. W. C. Reynolds. Berlin: Springer-Verlag, 1990.



Local instantaneous scalar dissipation rate in a vertical (x-z) plane for (left) homogeneous shear, and (right) isotropic flow.

Dispersion of Solid and Fluid Particles in Turbulent Homogeneous Flows With and Without Turbulence Modulation

S. E. Elghobashi, Principal Investigator
Co-investigator: G. C. Truesdell
University of California, Irvine

Research Objective

To study particle dispersion in turbulent isotropic and homogeneous shear flows and to examine the modulation of turbulence in heavily laden flows.

Approach

Direct numerical simulation is used to integrate the time-dependent three-dimensional Navier-Stokes equations to produce the velocity fields. A second-order Runge-Kutta scheme and fourth-order Hermitian polynomial interpolation are used to integrate the Lagrangian equation of particle motion. To study the two-way coupling between the particles and turbulence the two sets of equations are solved simultaneously.

Accomplishment Description

A simulation of Snyder and Lumley's experiment (1971) for particle dispersion in a decaying grid-generated turbulence was performed. Particle dispersion, diffusivity, and Eulerian and Lagrangian velocity autocorrelation coefficients for the fluid, particles, particles surrounding the fluid, and fluid points were just a few of the statistics computed. Good agreement with the experimental data was obtained. The figures show the dispersion curves and Lagrangian velocity autocorrelation coefficients for the simulation and for Snyder and Lumley's experiment. The dispersion computation needed about 82 megawords of memory and 7 Cray-2 hours for one run.

Significance

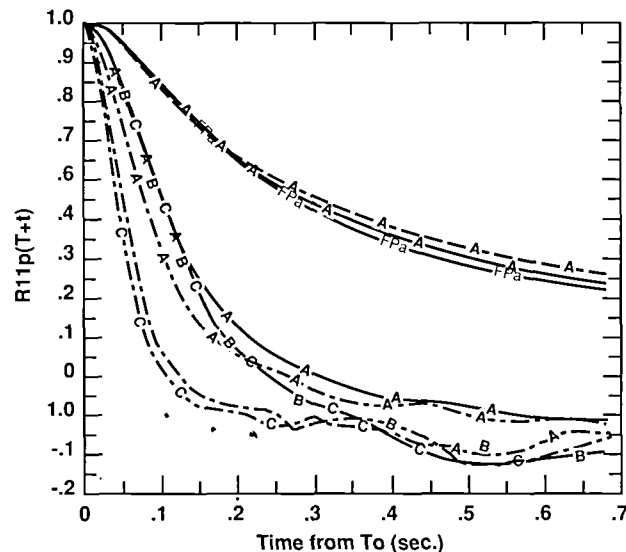
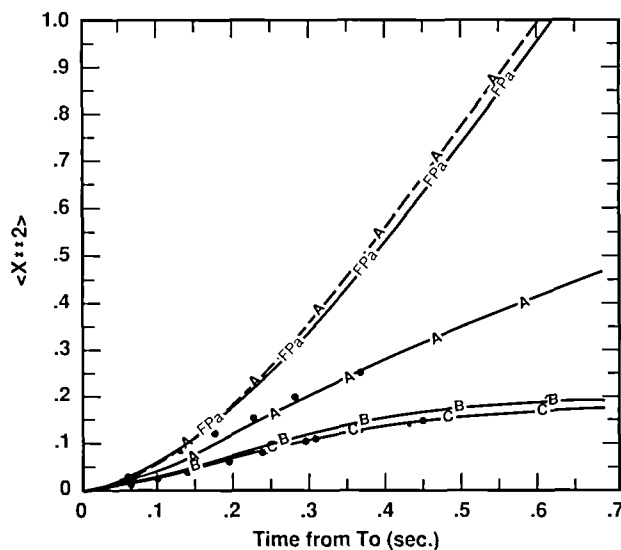
In a large number of mixing applications the mass and momentum of the particles are large enough to significantly alter the flow structure. Laboratory experiments designed to measure the modulation of turbulence in heavily laden flows are difficult to perform. The direct numerical simulation of particle/flow interaction is a numerical experiment that allows almost unlimited measurements of all details of the interaction, with relative ease.

Future Plans

Particle/flow interactions for heavily laden isotropic turbulence and homogeneous turbulent shear flows will be studied with various mass loadings and particle response times to determine their effects on the energy and structure of the flow and on particle dispersion statistics.

Publications

1. Elghobashi, S. E., and Truesdell, G. C. "Direct Simulation of Particle Dispersion in a Decaying Grid Turbulence." Presented at the Seventh Symposium on Turbulent Shear Flows, Stanford Univ., Palo Alto, CA, 1989.
2. Elghobashi, S. E., and Truesdell, G. C. "Direct Simulation of Particle Dispersion in Grid Turbulence and Homogeneous Shear Flow." *Bull. Am. Phys. Soc.* 34 (1989): 2311.



(Left) Normalized mean-square displacement. (Right) Lagrangian autocorrelation coefficients.

Control of the Flow Field about an Airfoil Section by Localized Surface Heating

N. M. El-Hady, Principal Investigator

Co-investigator: L. Maestrello

NASA Langley Research Center/Analytical Services and Materials, Inc.

Research Objective

This research uses localized surface heating to enhance the performance of an airfoil at high angles of attack. The objective is to investigate the flow field and assess its lift, flow separation, and stall characteristics with and without control.

Approach

The analysis is based on the numerical solution of the two-dimensional Reynolds-averaged compressible Navier-Stokes equations; an implicit, upwind-biased, finite-volume scheme with approximate factorization and flux difference splitting is used. Active control by heating is simulated by modifying the temperature boundary condition over a narrow strip located near the leading edge of the airfoil in the large-pressure-gradient region. A temperature control function with specified amplitude and frequency is designed to restructure the lift characteristics of the airfoil.

Accomplishment Description

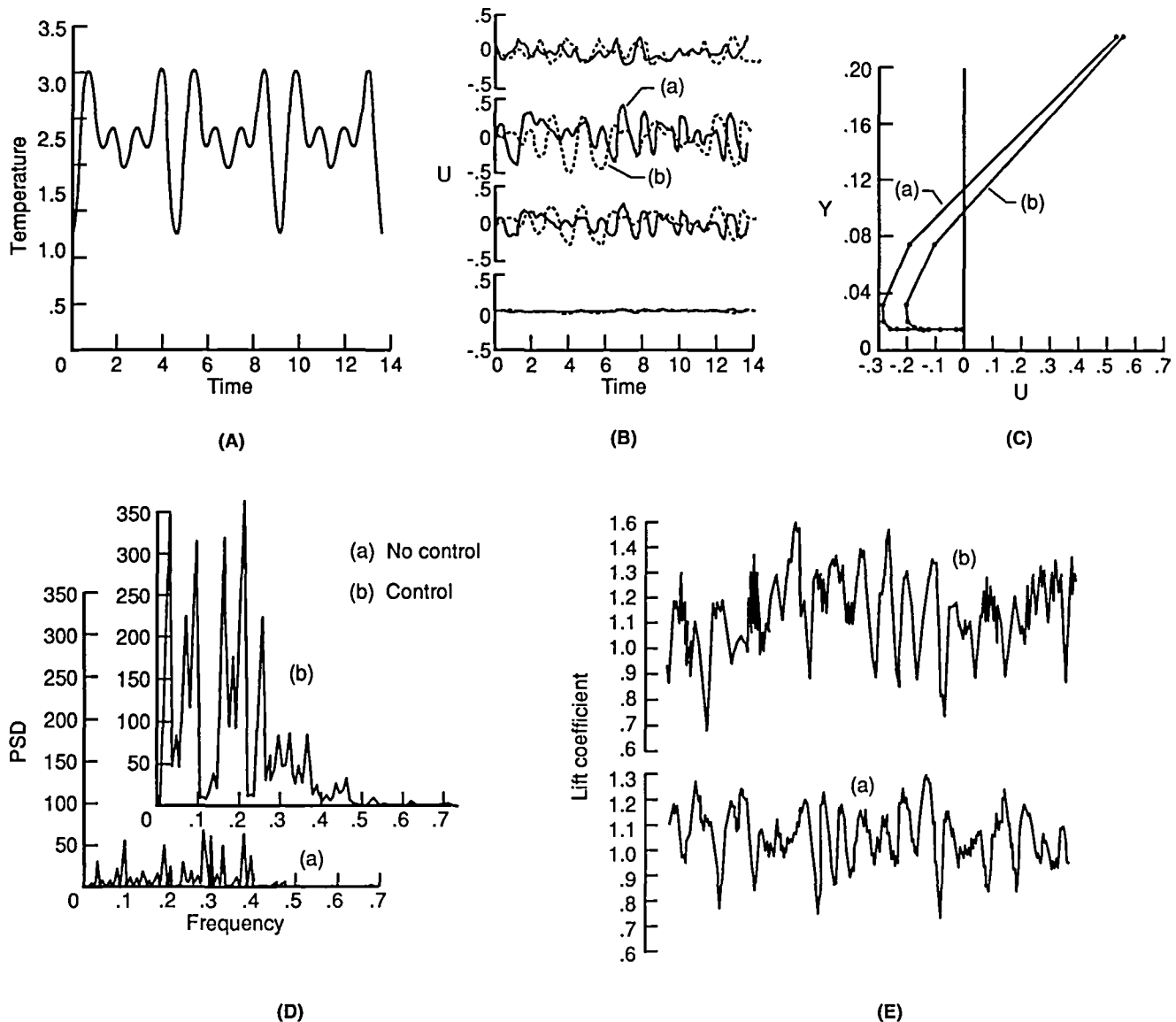
At $\alpha = 13^\circ$, the temperature function is tuned to the natural shedding frequencies of the shear layer to enhance the coupling between the surface heating and the separated flow. This leads to an increase in the rms value of the velocity perturbation, and a decrease in the size of the separated region. The power spectrum of the lift coefficient with control shows an amplitude that exceeds that of the uncontrolled lift. The lift and drag coefficients at this angle of attack oscillate around average values with an increase in the lift/drag ratio of 7%.

Significance

Separation control by localized surface heating can be used to improve the lift and stall characteristics of a maneuvering fixed-wing aircraft or of retreating blades on a helicopter rotor.

Future Plans

A parametric study is under way to evaluate the power required for and the practicality of the method, especially when it is applied to three-dimensional geometries.



(A) Temperature function. (B) Time history of velocity perturbation at $x/c = 0.42$ and different y locations. (C) Mean velocity profile near the surface at $x/c = 0.42$. (D) Lift coefficient frequency spectrum. (E) Lift time history.

CFD Analysis of the Flow Paths of Spherical Convergent Flap Nozzles

John J. Erhart, Principal Investigator
Co-investigators: Saadat A. Syed and Eric W. King
United Technologies, Pratt & Whitney

Research Objective

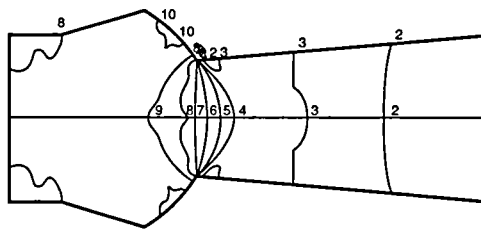
To provide analytical support in the development of advanced spherical convergent flap (SCF) nozzles.

Approach

A three-dimensional Navier-Stokes code with arbitrary curvilinear coordinates, k- ϵ turbulence model, and transonic flow capabilities was used to predict the flow field inside an SCF nozzle. An elliptic grid generator was used to create a mesh of the nozzle.

Accomplishment Description

The internal flow features of various configurations of the SCF nozzles were computed. These configurations showed the internal flow features inherent in the SCF nozzle, including turbulence, transonic flow, compression and expansion waves, boundary-layer skewness recirculation, and significant cross flow. Numerous calculations were made for two different SCF nozzles in straight, pitched, and pitched and yawed modes. These cases were run inviscid and viscous with various grid densities. The results were compared with pressure, thrust-coefficient, and vector-angle data.



LEGEND

- 1 0.1200E+02
- 2 0.1800E+02
- 3 0.2400E+02
- 4 0.3000E+02
- 5 0.3800E+02
- 6 0.4200E+02
- 7 0.4800E+02
- 8 0.5400E+02
- 9 0.6000E+02
- 10 0.6900E+02
- 11 0.7200E+02
- 12 0.7800E+02
- 13 0.8400E+02

Significance

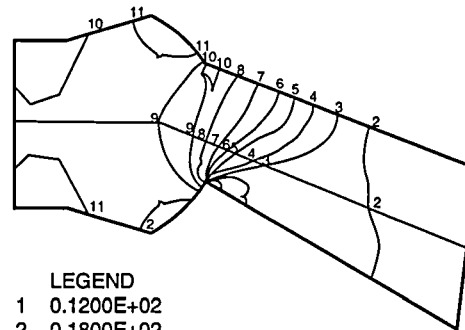
Exhaust nozzles for the next generation of fighter aircraft will be designed to have multiplane thrust-vectoring capability. Several studies have shown that the operational effectiveness of these aircraft can be enhanced by using advanced nozzles with thrust-vectoring/-reversing capabilities. Since the flow field generated inside these nozzles is highly three-dimensional and viscous, traditional design methods based on two-dimensional analyses are not adequate for their successful design. A need therefore exists to develop designs and design evaluation systems based on three-dimensional computational fluid dynamics methods that are accurate, economical, and have reasonably short turnaround time when applied.

Future Plans

The knowledge obtained from this work will be used to study the effect that cooling has on nozzle thrust.

Publications

Syed, S. A.; Erhart, J. J.; and King, E. W. "Application of CFD to Pitch/Yaw Thrust Vectoring Spherical Convergent Flap Nozzles." Presented at the AIAA/ASME/SAE/ASEE Joint Propulsion Conference, Orlando, FL, July 1990.



LEGEND

- 1 0.1200E+02
- 2 0.1800E+02
- 3 0.2400E+02
- 4 0.3000E+02
- 5 0.3800E+02
- 6 0.4200E+02
- 7 0.4800E+02
- 8 0.5400E+02
- 9 0.6000E+02
- 10 0.6900E+02
- 11 0.7200E+02
- 12 0.7800E+02
- 13 0.8400E+02

Pressure contours on nozzles used for aerocontrol; vectoring causes large flap-to-flap pressure gradients.

Compressible Turbulence

G. Erlebacher, Principal Investigator

Co-investigators: S. Sarkar, C. Shu, and T. A. Zang
NASA Langley Research Center/ICASE

Research Objective

To study the physics of compressible homogeneous turbulence, and to generate accurate data bases against which turbulence models can be evaluated.

Approach

The approach was to solve the time-dependent Navier-Stokes equations with constant viscosity and conductivity in a periodic three-dimensional box. Theory and numerical diagnostics were used to distinguish the compressible from the incompressible flow characteristics.

Accomplishment Description

A theoretical framework was developed to predict the possible states of isotropic, decaying, compressible turbulence as a function of initial conditions. Formulas that predict the equilibrium levels of compressibility, dissipation rates, and dilatational variances were established and checked against the results of two- and three-dimensional direct numerical simulations (DNS). The key result was that in the absence of shocks, the turbulence evolves toward a state of equipartition of the compressible components of potential and kinetic energy. From theoretical considerations, turbulent models were developed to treat both the pressure dilatation and the compressible dissipation terms in the energy equation. These models, when incorporated into a second-order closure model, predicted the correct convective Mach number (M_c) dependence of the mixing rate in a compressible mixing layer. This can be seen in the figure (taken from Publication 1), which shows the spreading rate of a compressible shear layer as a function of M_c . Three-dimensional DNSs were conducted on 64^3 grids to calibrate the model constant. Each required about 5 CPU hours. An essentially nonoscillatory third-order scheme for two- and three-dimensional flows was also developed and applied to the simulation of compressible turbulence. The data generated were compared with DNS data and were found to predict accurately shock locations and speeds. A total of approximately 400 CPU hours and 200 deferred hours were used during the March 1989 to March 1990 operational period.

Significance

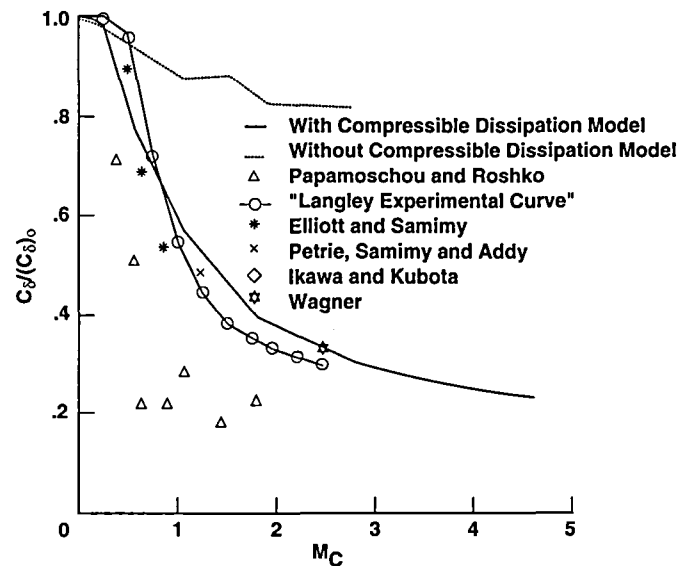
Two- and three-dimensional direct simulations, properly coupled with experiments and theory, improve our understanding of turbulence and provide a good foundation on which to build turbulence models.

Future Plans

Two- and three-dimensional simulations of turbulent shear flows will be performed to assess whether the presence of weak curved shocks enhances mixing when the convective Mach number is supersonic.

Publications

1. Sarkar, S., and Balakrishnan, L. ICASE Report No. 90-18.
2. Erlebacher, G.; Hussaini, M. Y.; Kreiss, H. O.; and Sarkar, S. ICASE Report No. 90-15.
3. Sarkar, S.; Erlebacher, G.; Hussaini, M. Y.; and Kreiss, H. O. ICASE Report No. 89-79.



The spreading rate of a compressible shear layer.

Three-Dimensional Viscous Drag Prediction for Rotor Blades

Fort F. Felker, Principal Investigator
Co-investigator: Ching S. Chen
NASA Ames Research Center

Research Objective

To provide an accurate and efficient design tool to the rotorcraft industry to predict the rotor-blade drag force for rotorcraft in hover and forward flight.

Approach

A viscous-inviscid interaction approach is used. Very near the blade surface, the flow is assumed viscous and is governed by the three-dimensional boundary-layer equations. Outside the boundary layer, the flow is assumed inviscid and is governed by the full-potential equations. These two sets of equations are solved separately on a rotating reference frame but are coupled together through pressure and displacement thickness.

Accomplishment Description

The development of the boundary-layer code in the rotating reference frame and the coupling of this code with an existing transonic full-potential code were completed. The accuracy of the numerical scheme was validated by comparing computed results with theoretical results and test data for laminar and turbulent flow over a flat plate and for nonlifting two-dimensional airfoil flows over a range of Mach numbers. The method was recently applied to three sets of nonlifting rotor flows, two in hover and one in forward flight. The results demonstrate that the current analysis is able to predict the nonlifting rotor drag for general rotor configurations. The comparison of computed torques with test data for the two

hovering rotors are shown in the accompanying figure. Each rotor flow computation required approximately 1 hour of Cray-2 time and 8 mega-words of memory.

Significance

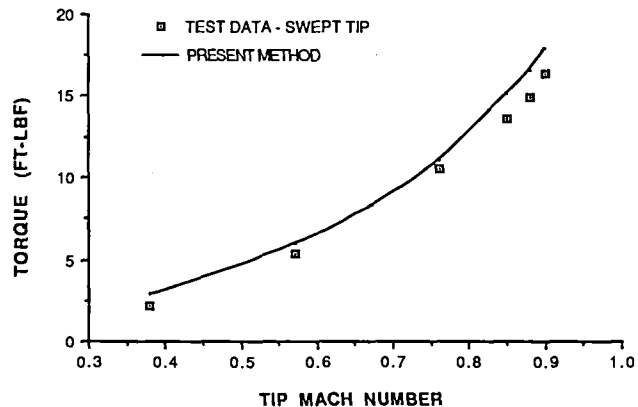
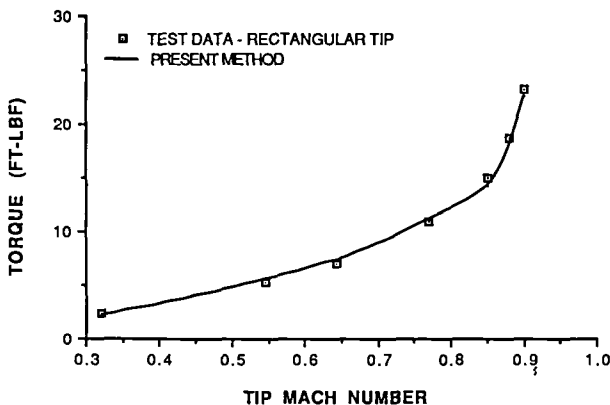
The ability to accurately predict drag force for rotating blades is extremely important in the design of rotorcraft. The state of the art in rotor-blade drag prediction involves the use of two-dimensional airfoil tables to extrapolate the drag force on the blade. This method does not take the Reynolds number or the rotational effects of the blade into account, raising doubts about the accuracy of the results. This project addresses this problem with the development of an analytical method that includes the effects of Reynolds number as well as the rotational motion of the blade. If successful, it is anticipated that this analysis will be used extensively in the design of future rotorcraft.

Future Plans

The codes will be enhanced to allow simulation of lifting rotor flows as well as nonlifting flows. In addition, the coupling of the current analysis with a rotor wake and blade dynamics model will be completed.

Publications

Chen, C. S., and Bridgeman, J. O. "Three-Dimensional Viscous Rotor Flow Calculations Using Boundary-Layer Equations." Presented at the Fifteenth European Rotorcraft Forum, Amsterdam, The Netherlands, Sept. 1989.



Comparison of computed torques with test data for a nonlifting hovering rotor with (left) rectangular-tip blades, and (right) swept-tip blades.

Compressible Navier-Stokes Code Development

Jolen Flores, Principal Investigator
Co-investigators: Steve Ryan and Tom Edwards
NASA Ames Research Center

Research Objective

To develop the ability to predict the hypersonic, viscous flow over a complicated geometry; perfect, equilibrium, and nonequilibrium flows are included.

Approach

A three-dimensional Navier-Stokes code is implemented in a zonal approach, with options for perfect-gas, equilibrium, and nonequilibrium flow. The compressible Navier-Stokes (CNS) code is a zonal flow solver for use in hypersonic vehicle design. It contains a version of the F3D, partially flux-split, thin-layer Navier-Stokes flow solver. That solver is applied using a zonal approach derived from the transonic Navier-Stokes (TNS) code.

Accomplishment Description

The Cray-2/Cray Y-MP time allotted to the CNS project has been used to develop the finite-rate air-chemistry option now available in the CNS version 1.4. Development of the code has benefited from use of the NAS facilities. Some of the accomplishments of the project are described below. First, a perfect-gas solution over the McDonnell-Douglas generic-option blended wing-body was computed. The purposes of this exercise were to check out the zonal grid interfacing capabilities, to compare the CNS results with those of a space-marching method, and to create a data base for future code enhancements. The figure on the left shows the grid for the generic option, which consists of eight zones totaling over

350,000 points. The flow conditions for this test case were $M_\infty = 11.35$, $\alpha = 0^\circ$, and $Re = 31 \times 10^6 \text{ m}^{-1}$. Results were compared with experimental data and a parabolized Navier-Stokes calculation. Comparisons for the windward centerline surface pressure are shown in the figure on the right. This computation required about 30 hours of CPU time and about 10 megawords of memory. Second, the finite-rate air option was exercised over a sphere for a variety of Mach numbers. These same cases were run for a two-zone sphere case.

Significance

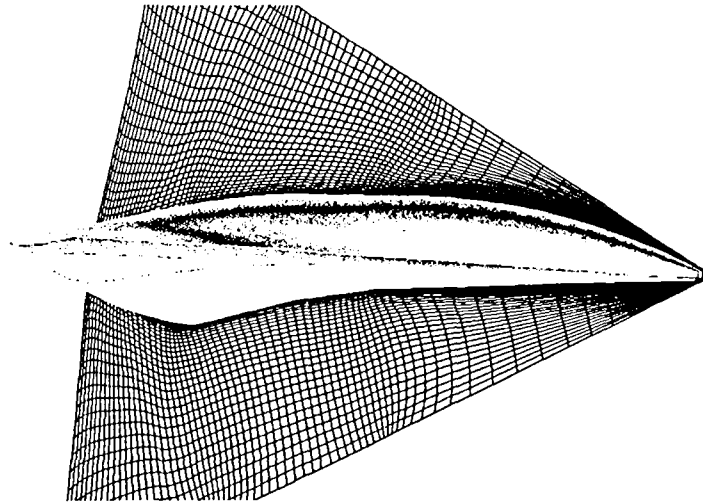
The CNS code is now ready for detailed validation, for application to complicated vehicles, and for integration with an interflow code to allow complete nose-to-tail hypersonic analyses.

Future Plans

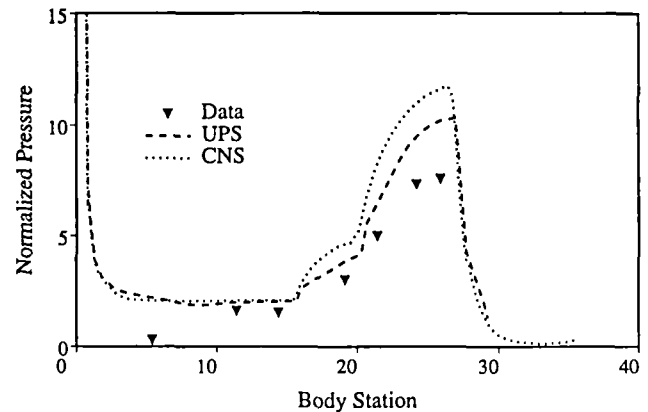
The finite-rate version of the CNS code will now be applied, in conjunction with the UPS code, to more complicated geometries. Sensitivity studies will be conducted to analyze the effects of nonequilibrium flow on pressure and heat-transfer values.

Publications

Lawrence, S., and Flores, J. "Air Chemistry Sensitivity Analysis Using the CNS/UPS Codes." Presented at the Eighth National Aerospace Plane Technology Symposium, Monterey, CA, Mar. 1990.



(Left) Geometry and grid for the generic-option aerothermal model.



(Right) Pressure comparisons on windward symmetry plane of generic-option model at $M_\infty = 11.35$ and $\alpha = 0^\circ$.

Electronic-Structure Study of High Superconductors

Arthur J. Freeman, Principal Investigator

Co-investigators: A. Continenza, S. Massidda, and J. Yu
Northwestern University

Research Objective

To carry out electronic-structure calculations on the new high-temperature superconductors in order to gain an understanding of their electronic structure and properties, as well as to provide insight into the mechanism of superconductivity of these high T_c materials.

Approach

The highly precise, full-potential, linearized, augmented plane-wave method is used to solve the time-independent quantum mechanical equations in the local density approximation.

Accomplishment Description

Recently, the discovery of the so-called electron-doped superconductors (e.g., $\text{Nd}_{2-x}\text{Ce}_x\text{CuO}_4$) added a new dimension to the theoretical model for the mechanism of high T_c superconductivity. The results of our highly precise, local-density electronic-structure calculations on Nd_2CuO_4 showed its close similarity to the "hole-doped" superconductors. (The similarities are the important role of the strongly two-dimensional Cu-O antibonding $dp\sigma$ band, a somewhat lower density of states at the Fermi energy, and a highly two-dimensional Fermi surface.) This symmetry in electron-hole doping appears to be additional evidence for the validity of a Fermi liquid description of the normal metallic state of the high T_c superconductors. An average single run of the calculations, which solve secular determinants of the order 1200×1200 many times in the (iterative) self-consistency procedure, uses about 100 minutes of Cray-2 time and requires about 16 megawords of memory.

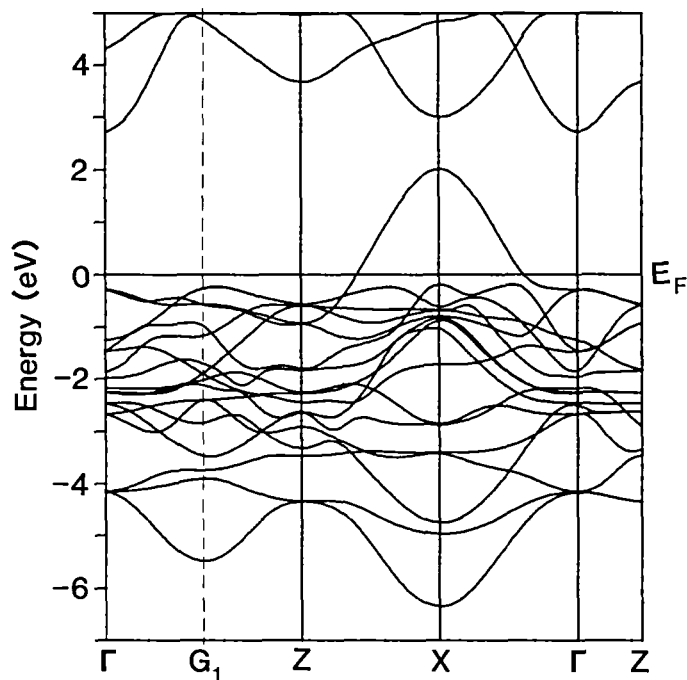
Significance

For years there has been controversy and confusion among theorists as well as experimentalists about whether the "normal" state of the Cu-oxide superconductors is a Fermi liquid or some other exotic ground state. Recent experimental observations have confirmed the predictions of our local-density functional calculations based on the Fermi liquid picture; hence the Fermi liquid (metallic) nature of the "normal" state of the high T_c superconductors has been established.

Future Plans

A primary goal will be to develop a microscopic picture for the electron-electron interactions in high T_c Cu-oxide super-

conductors, based on our energy band results. In particular, we are interested in the dielectric response, which provides crucial information for many dynamical properties, including structural and magnetic transitions and electron-phonon interactions as well as superconductivity in high T_c superconductors.



Calculated energy band structure of Nd_2CuO_4 along the main symmetry directions in the Brillouin zone. The single band crossing Fermi level E_F is the strongly two-dimensional Cu-O antibonding $dp\sigma$ band, which is common with the other high T_c Cu-O superconductors.

Computational Studies of Compressibility Effects on Dynamic Stall

K.-Y. Fung, Principal Investigator
University of Arizona

Research Objective

To identify the onset mechanisms in dynamic stall.

Approach

A series of computations was carried out using a Reynolds-averaged Navier-Stokes code. Two airfoils, the NACA 0012 and the VR7, were chosen for this study. For each airfoil and Mach number, the flows that correspond to different angles of attack were obtained. Results of the computations were analyzed and compared with available experimental data.

Accomplishment Description

It was found that the onset of stall can be delayed by increasing the frequency of oscillation, as long as the flow remains subsonic. Once the flow is locally supersonic, the onset of stall becomes much less sensitive to increased frequency but has a strong dependence on the free-stream Mach number. The onset's dependence on the Mach number is not affected by the airfoil geometry as much as is its dependence on the reduced frequency. Before the onset of stall, the instantaneous pressure distributions over the airfoil can be considered quasi-steady and can be predicted using inviscid theory. A boundary-layer analysis, using computed inviscid flows and classical theory for static stall, suggests that once the flow becomes locally supersonic, the onset of stall is a result of the interaction between the forming shock and the steepening laminar boundary layer. It also explains the differences between the onset characteristics of the NACA 0012 and VR7

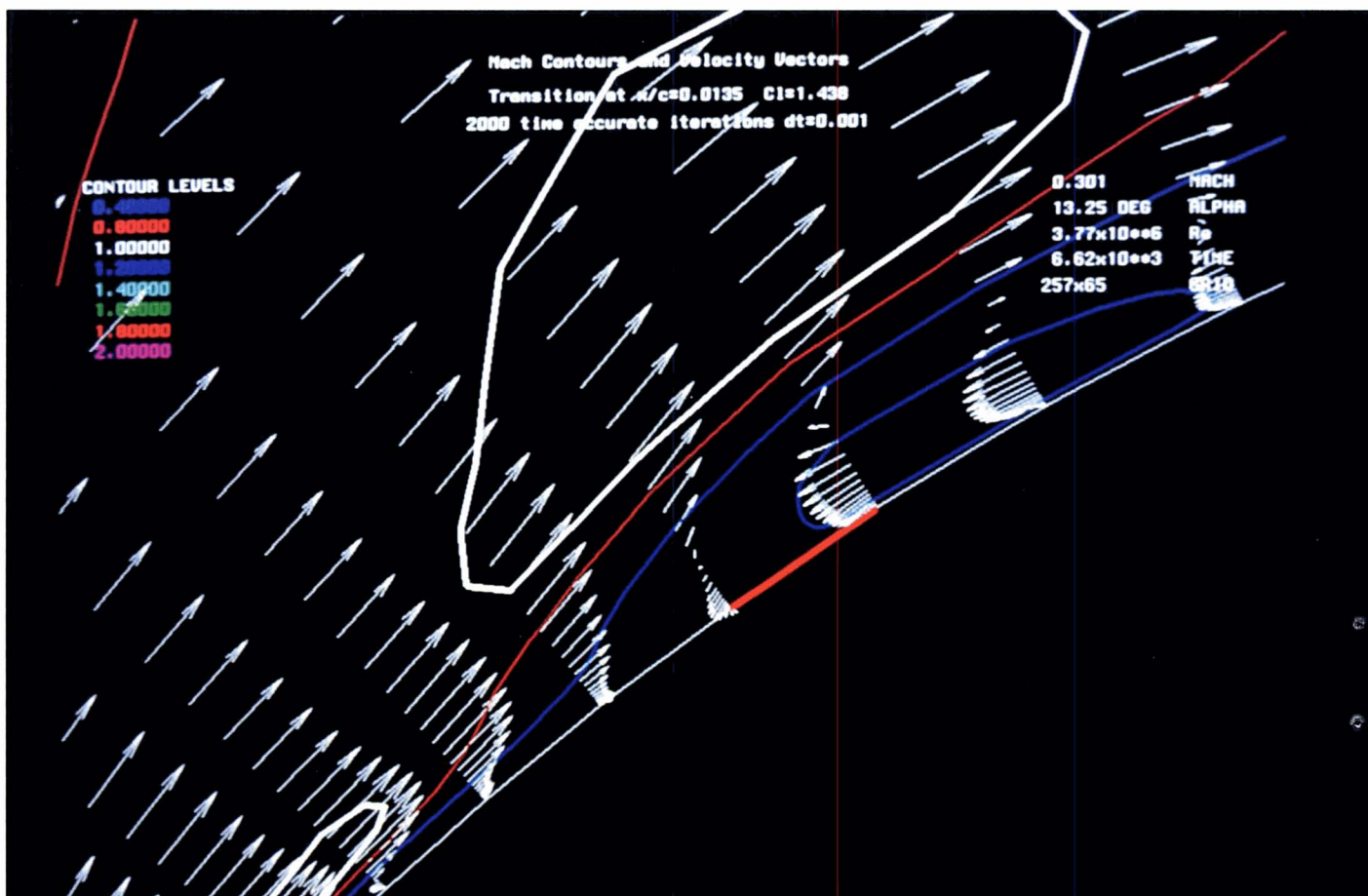
airfoils. The sensitivity of stall onset on transition was studied by computing the flow over an airfoil at conditions near stall and varying the switch-on location of the turbulence model for the Reynolds-averaged Navier-Stokes code used in this study. It was found that the flow is not sensitive to the switch-on location when the angle of attack is below the static stall value. For an angle of attack close to the static stall value, e.g., at 13.25° , a mere change of transition locations from 1.25% to 1.35%, which differ by one grid point and are before and after the computed shock location respectively, causes the flow from reattachment to massive separation (see figure). For a lower free-stream Mach number, 0.185, separation is less sensitive to the transition location as the angle of attack is increased.

Significance

Some insight into the onset of dynamic stall has been gained from our work. Before the onset, the validity of quasi-steady inviscid theory is confirmed. The dependence of the onset of stall on the Mach number and leading-edge profile can be explained by classical boundary-layer theory. The importance of separation bubble bursting and transition effects on reattachment deserve a renewed assessment.

Future Plans

Numerical experiments will be designed to study the stability of reattaching separated flows at dynamic stall conditions.



Mach contours and velocity vectors.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Validation of 3-D Wind Tunnel Wall Interference Assessment/Correction Codes

Javier A. Garriz, Principal Investigator
NASA Langley Research Center

Research Objective

To develop the capability to assess the extent to which wind tunnel data can be corrected (by adjustments to the free-stream Mach number and configuration angles of attack) to account for the effects of the tunnel walls, sting mounts, and other tunnel environment parameters.

Approach

To obtain corrections for both Mach number and angle of attack, a three-dimensional small-disturbance code (with nonisentropic corrections), CAP-TSD, is used to solve two boundary-value problems. First, an "in-tunnel" boundary-value problem is solved using measured tunnel-wall data as outer boundary conditions, subject to matching measured lift and pitching moments. Then, a "free-air" boundary-value problem is solved in which the free-air Mach number and angle(s) of attack are varied so as to minimize the rms error between the in-tunnel and free-air model surface Mach numbers, subject to matching the total lift.

Accomplishment Description

The three-dimensional small-disturbance code CAP-TSD has been adapted for use as a wall interference assessment/correction code. The code has been used to assess the interference present in several three-dimensional data sets involving full-span and semispan tunnel models. The resulting

corrections have been verified (at least for semispan models) by an independent, free-air thin-layer Navier-Stokes solver. The correction code required an average of 24 megawords and 2 Cray-2 hours, and the independent free-air solver required 40 megawords and 4 Cray-2 hours.

Significance

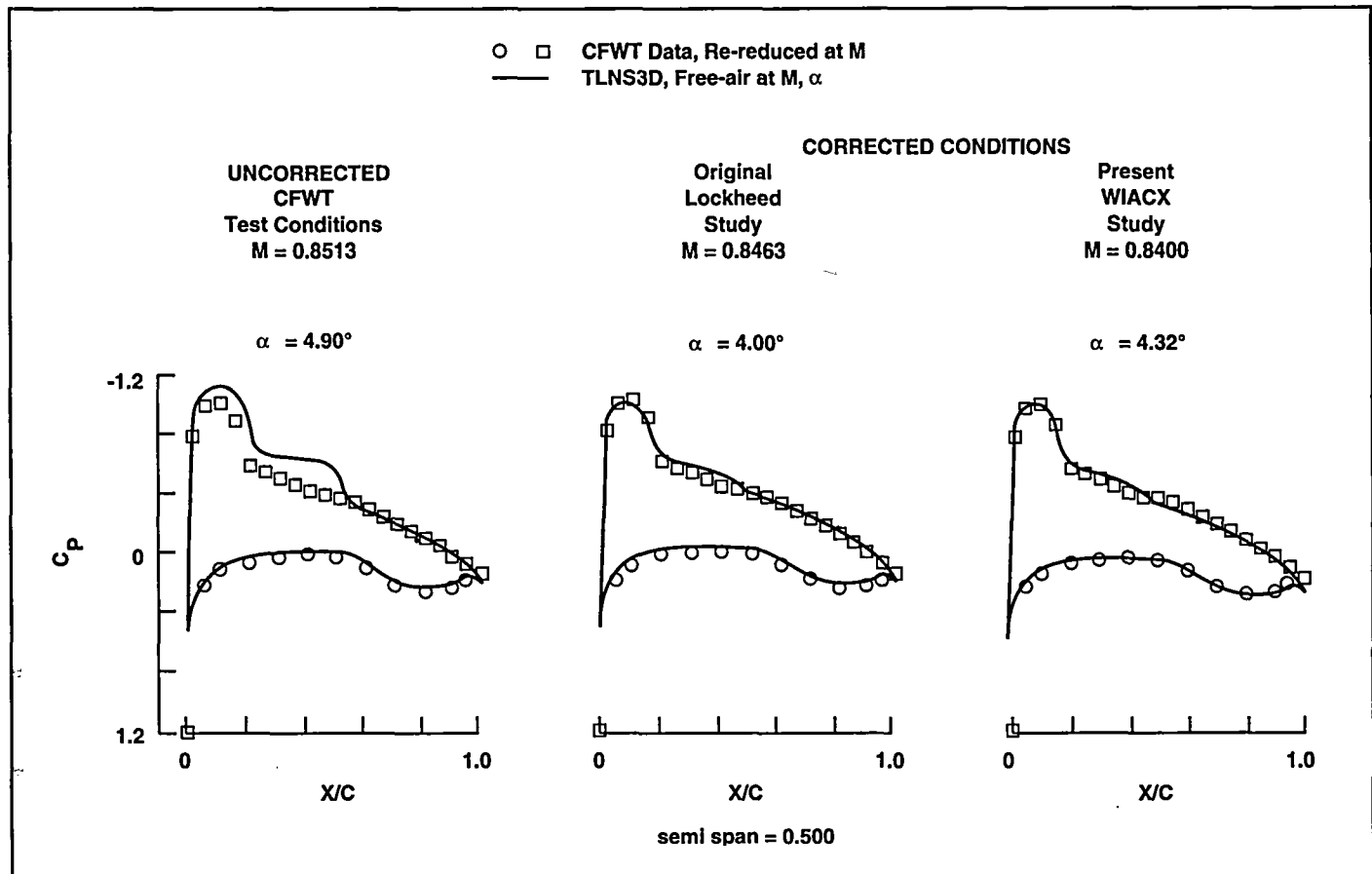
The wall interference for models tested in transonic wind tunnels must be assessed and perhaps corrected in order to meet current data accuracy demands. The need to determine the free-air conditions that should be associated with the measured tunnel data is shared by anyone interested in using such data, including those who use it for comparison with or validation of other computational fluid dynamics codes.

Future Plans

The code will be used on additional three-dimensional data sets to continue the validation process, including data sets from the National Transonic Facility, the tunnel for which the code was originally developed.

Publications

Garriz, Javier A.; Newman, Perry A.; Vatsa, Veer N.; Haigler, Kara J.; and Burdges, Kenneth P. "Evaluation of Transonic Wall Interference Assessment and Corrections for Semi-Span Wing Data." Presented at the AIAA 16th Aerodynamic Ground Testing Conference, Seattle, WA, June 1990.



Comparison of results for Lockheed wing C data from CFWT at $Re_{mac} = 10.36 \times 10^6$ and 4% porosity.

Numerical Simulations of Deep Global Convection in Jupiter

Gary A. Glatzmaier, Principal Investigator
Los Alamos National Laboratory

Research Objective

The overall objective is to provide a detailed, self-consistent physical explanation for the maintenance of the banded zonal wind pattern observed on Jupiter by the Voyager spacecraft. A successful numerical simulation would also provide predictive descriptions of the three-dimensional velocity, magnetic field, and thermodynamic structures and time dependencies throughout the entire interior of Jupiter.

Approach

A computer code solves an equation of state, a mass conservation equation, a momentum equation, a heat equation, a magnetic-flux conservation equation, and a magnetic induction equation to simulate time-dependent, three-dimensional, compressible convection and magnetic field generation in a stratified, rotating, spherical fluid shell. The anelastic approximation is used to filter out sound waves. The horizontal structures of the velocity, thermodynamic, and magnetic fields are represented by spherical harmonic expansions, and the radial structures by Chebyshev polynomial expansions. A spectral transform method computes, at every time step, all spatial derivatives analytically in spectral space and all nonlinear terms in grid space. Over one million grid points are used in the three-dimensional spherical shell. The time integration treats the linear terms implicitly and the nonlinear terms explicitly.

Accomplishment Description

The original model generated a banded zonal velocity field that, except in the equatorial region, was in fair agreement

with the Voyager observations (see figure in "NAS Technical Summaries" Mar. 1988–Feb. 1989). The failure of the model to generate the observed equatorial jet was apparently due to the large eddy viscosity required by the model. A new version of the model that uses a different numerical method is now being tested. The accompanying figure shows a snapshot of temperatures and velocities in the equatorial plane for a case that tests some of the new algorithms. (This case does not use the rotation rate of Jupiter.) Reds represent warm fluid rising; blues represent cold fluid sinking. The arrows represent the direction and relative magnitude of the fluid velocity in this plane through the sphere. Each test requires many 4-hour, 10-megaword runs on the Cray-2.

Significance

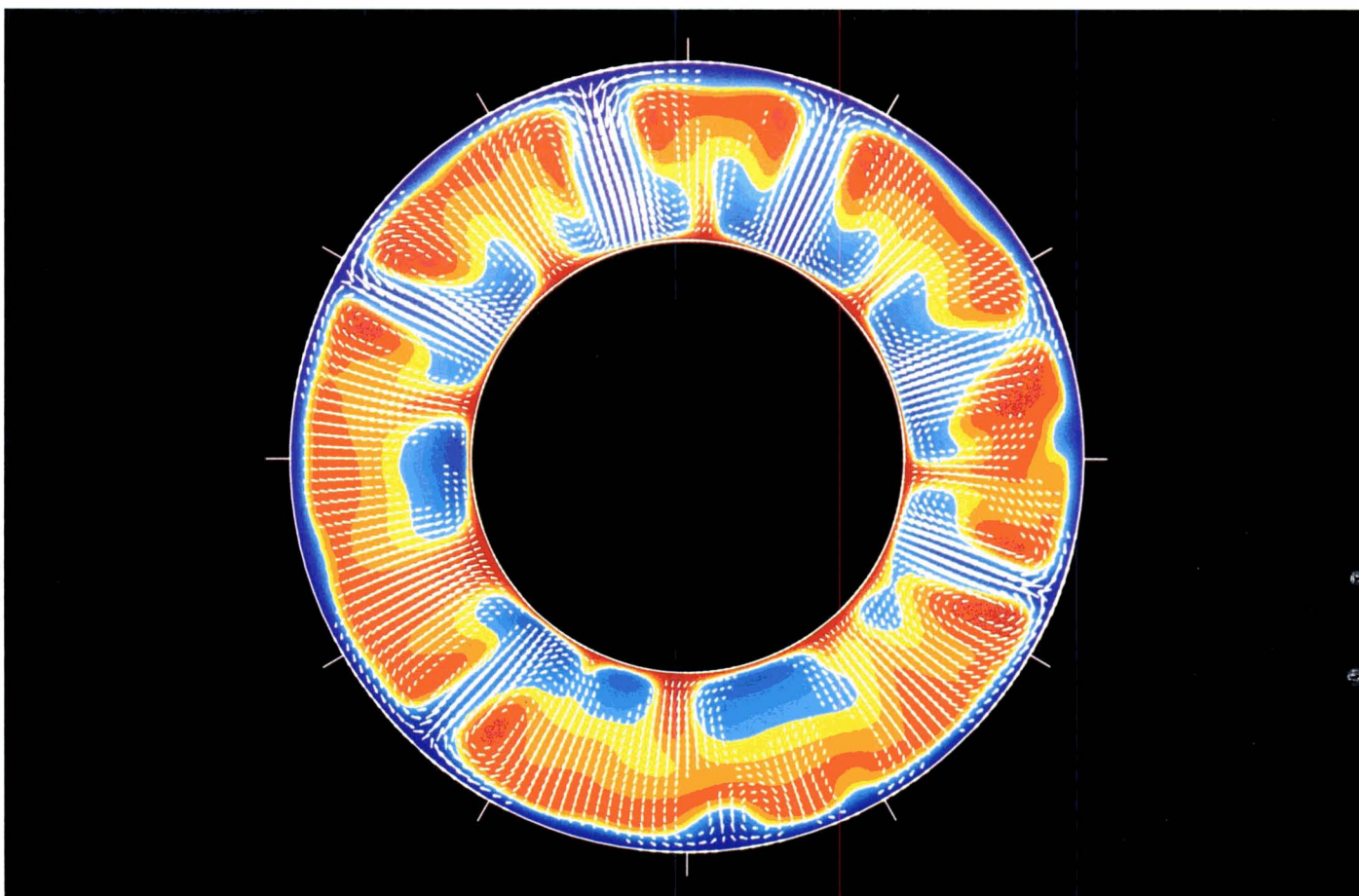
Although Jupiter's banded zonal wind pattern has been accurately measured, there are presently no self-consistent physical explanations for why it exists or how deep below the surface it persists. A detailed analysis of the forces responsible for maintaining such a pattern in a successful numerical simulation would provide these explanations.

Future Plans

Improvements will continue to be made in the model. When the simulation of Jupiter is satisfactory, the other giant planets will be simulated.

Publications

Glatzmaier, Gary A. "3D Global Numerical Simulations of Convection and Magnetic Field Generation in Jupiter." *Bull. American Astronomical Soc.* 21 (1989): 752.



Simulation of temperatures and velocities in the equatorial plane of Jupiter.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Hypersonic Flows in Chemical and Thermal Nonequilibrium

Peter A. Gnoffo, Principal Investigator
NASA Langley Research Center

Research Objective

To make possible the simulation of hypersonic flow fields in chemical and thermal nonequilibrium over realistic geometries, with particular emphasis on Aeroassisted Orbital Transfer Vehicle and Mars-return applications.

Approach

The three-dimensional Navier-Stokes equations, 11 species continuity equations, a total energy equation, and a vibrational energy equation are solved in a fully coupled, point implicit, symmetric-total-variation-diminishing algorithm.

Accomplishment Description

Aeroassist Flight Experiment (AFE) trajectory points at three altitudes and two or three angles of attack per point were computed and used to define aerodynamic coefficients of the complete configuration. Generally good comparisons were obtained with several ground-based and in-flight experiments, including Project FIRE (heat transfer), RAM-C (electron number density), shock tunnel (interferogram), ballistic range (shock shape), and Mach 10 wind tunnel (pressure and heat transfer). The first four tests made possible a limited evaluation of the nonequilibrium physics modeled in the code. The last test provided initial confirmation of the ability of the code to predict shear-layer impingement heating on the payload behind an aerobrake. Impingement-heating studies on the AFE carrier are ongoing, with emphasis placed on the worst-case scenario at a -5° angle of attack (see accompanying figure). Finite-catalytic-wall boundary conditions were added to the code. Preliminary studies were completed that used reduced chemical kinetic models, as defined by the computational singular perturbation (CSP) method for the AFE shock layer, to enhance code efficiency in the presence of complex

chemical kinetic models. A preliminary evaluation of the code for higher velocity entries characteristic of return from Mars was prepared.

Significance

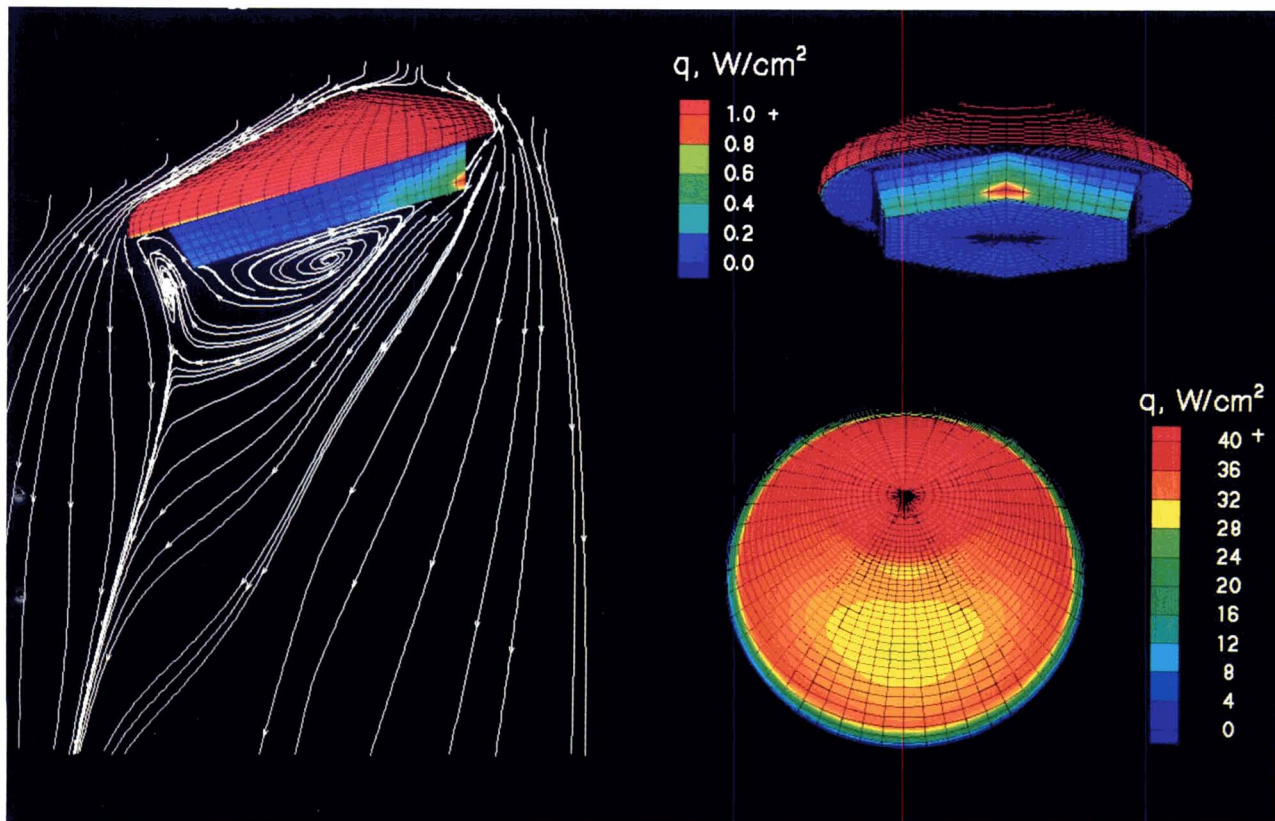
The present algorithm, coupled with improved physical models, will be used to benchmark design future Aeroassisted Orbital Transfer Vehicles, lunar- and Mars-return aerobrakes, and aerospace planes. It is now being used in the design of the AFE vehicle, scheduled for flight in 1994.

Future Plans

The code will be restructured to permit more efficient use of parallel processing and to facilitate the implementation of the CSP procedure. The matrix of cases for the AFE flight project will be updated to include the effects of finite-rate wall catalysis, with emphasis on heating predictions on the forebody and impingement points on the carrier. Parameter studies of kinetic models and their effects on uncoupled, nonequilibrium radiation and electron number density will be conducted in support of AFE flight experiments.

Publications

1. Gnoffo, Peter A. "Upwind-Biased, Point-Implicit Relaxation Strategies for Viscous, Hypersonics Flows." AIAA Paper 89-1972-CP, 1989.
2. Gnoffo, Peter A. "An Upwind-Biased, Point-Implicit Relaxation Algorithm for Viscous, Compressible Perfect-Gas Flows." NASA TP-2953, Feb. 1990.
3. Gnoffo, Peter A. "A Code Calibration Program in Support of the Aeroassist Flight Experiment." *J. Spacecraft and Rockets* 27, no. 2 (Mar.-Apr. 1990).



AFE surface heating; $V_\infty = 9.3 \text{ km/s}$, $h = 75 \text{ km}$, $\alpha = -5^\circ$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Flow-Field Simulation Around Different Rotor Planforms

Robert A. Golub, Principal Investigator

Co-investigator: Forooz F. Badavi

NASA Langley Research Center/Lockheed Engineering and Sciences Company

Research Objective

To develop computational methods for the aerodynamic and aeroacoustic comparison of transonic flow around different rotor planforms.

Approach

The transonic flow around a helicopter rotor in forward flight is calculated on a body-conformed grid around a rectangular and a swept-back rotor planform by coupling a recently developed three-dimensional Navier-Stokes solver with the wake analysis code CAMRAD. The Navier-Stokes solver is developed using an implicit, upwind-biased, finite-volume scheme that uses the Van Leer flux vector splitting and Beam-Warming approximate factorization techniques.

Accomplishment Description

Detailed profiles were obtained of surface pressure as a function of azimuth at different chord locations, and of local Mach numbers at the leading edges of rotors as a function of radial location at different azimuths. Comparisons of these profiles indicate that the performance of the swept-back rotor

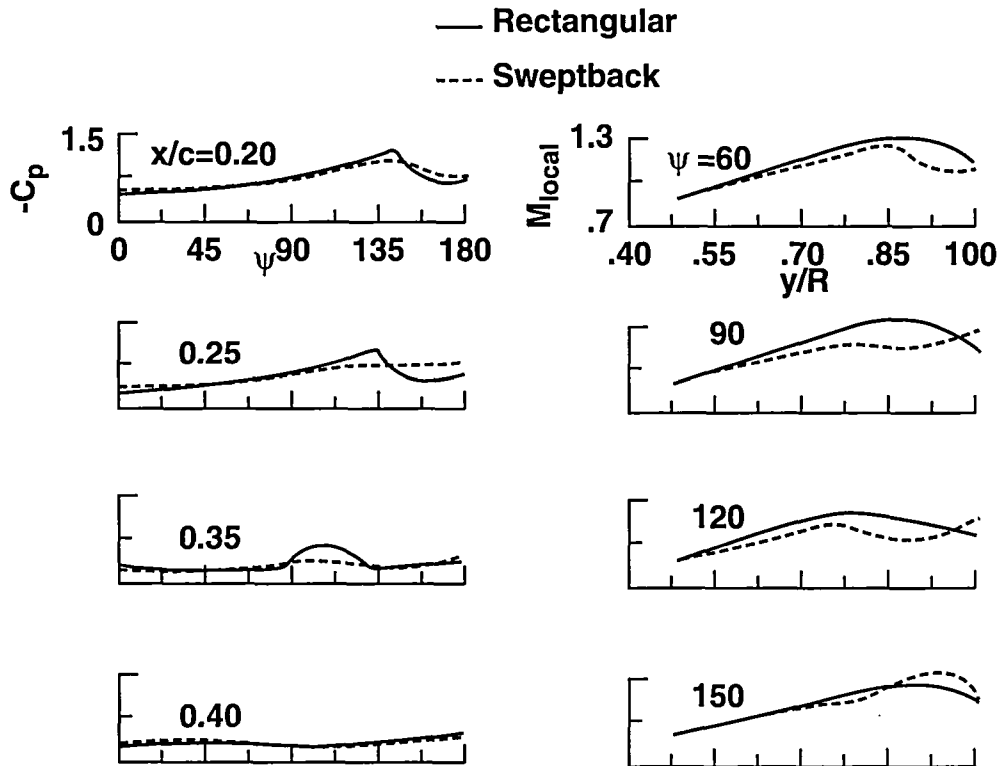
is better than that of the rectangular rotor at transonic tip Mach numbers. These comparisons show that the primary contribution of the swept-back rotor is the delay of shock-induced separation. The figures on the left compare the surface pressures of two rotors, and the figures on the right compare the leading-edge local Mach numbers.

Significance

With the current push toward high-speed-rotorcraft research, the present algorithm, with its demonstrated capability, supplies a method for studying either the global or detailed boundary-layer features of different rotor planforms at transonic tip Mach numbers.

Future Plans

The future plans for the present algorithm are to make the grid topology more efficient, to extend the spatial accuracy of the scheme to higher order to resolve the attenuation of propagating acoustics waves in the far field, and to implement a more accurate turbulence model.



Correlation of numerical results for nonlifting rotors in forward flight; $M_t = 0.630$, $\mu = 0.39$. (Left) Surface pressure. (Right) Leading-edge local Mach number.

A Supersonic Computational Fluid Dynamic Analysis of a Generic Fighter

Aga M. Goodsell, Principal Investigator
NASA Ames Research Center

Research Objective

To evaluate the solutions, obtained from an Euler code, of supersonic flow fields about a generic fighter, in order to determine their validity over a wide range of angles of attack.

Approach

The Euler computations are performed using FLO57, a three-dimensional finite-volume code, originally written by Antony Jameson and modified by several users in order to accommodate an O-H grid topology. Computed surface-pressure distributions, forces, and moments are compared to experimental data.

Accomplishment Description

The FLO57 solutions were obtained at Mach numbers of 1.2, 1.4, and 1.6 for angles of attack between 4° and 20°. Two configurations were studied, a wing/body and a wing/body/chine. The FLO57 code predicted pressure distributions, forces, and moments well at low angles of attack, at which the flow was fully attached. The agreement between Euler and experimental results were not as good when the vortex was just beginning to form, but the agreement improved when the vortex was well established. The addition of the chine did not create a marked change in the pressure distributions on the wing. The main effect of the chine was to push the leading-edge vortex outboard.

Significance

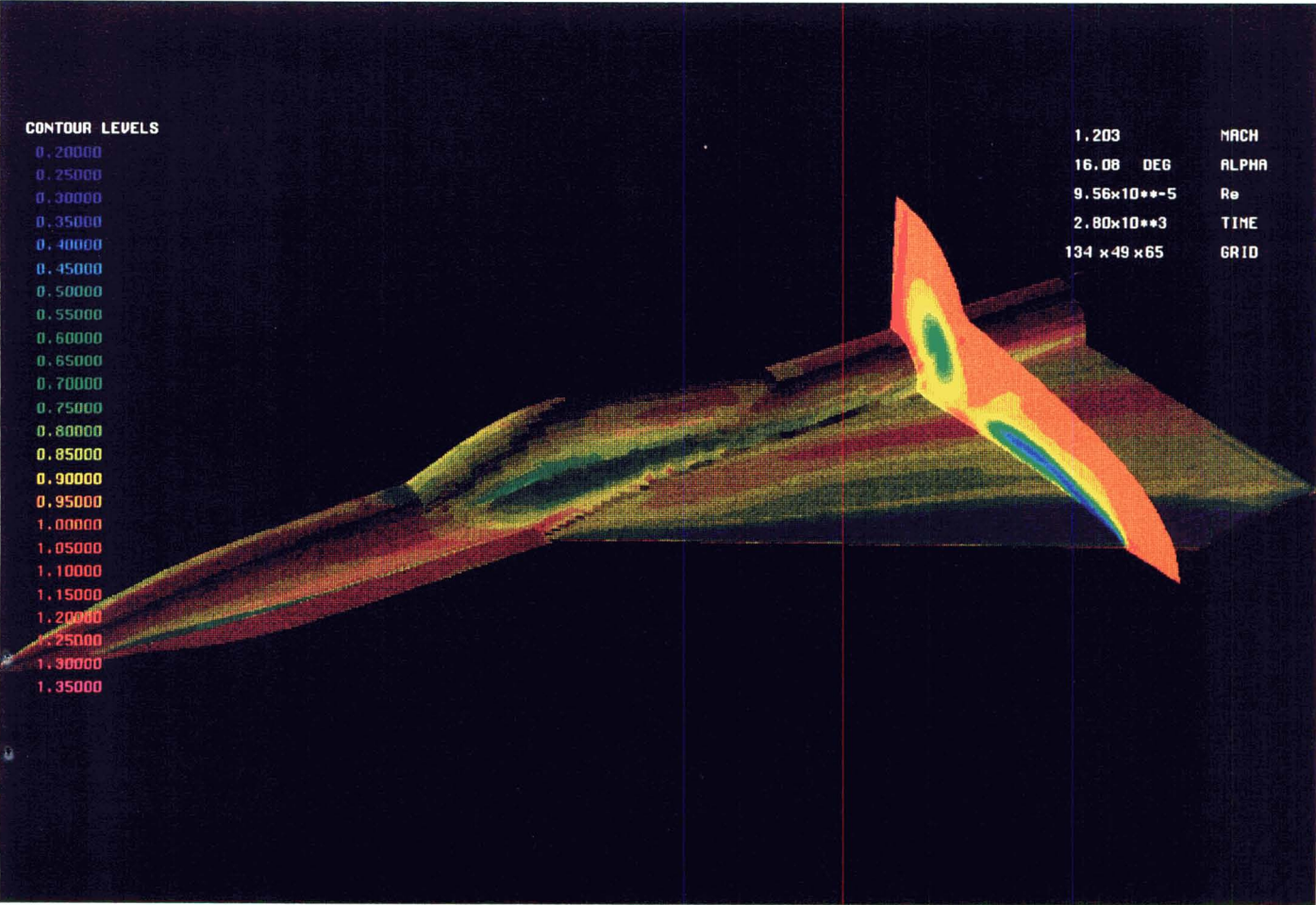
The validity of flow solutions about complex configurations is critical to the successful computational analysis of current and future aircraft geometries. The only method of determining the validity is to compute and analyze numerous solutions over a broad range of conditions and compare them with experimental results.

Future Plans

Navier-Stokes computations using LANS3D will be obtained. First the wing/body configuration will be analyzed; then the computations will include the wing/body/chine.

Publications

1. Goodsell, Aga M.; Madson, Michael D.; and Melton, John E. "TranAir and Euler Computations of a Generic Fighter Including Comparisons with Experimental Data." AIAA Paper 89-0263, Jan. 1989.
2. "Visualization of Vortex Flows about Chine-Delta Configurations." Video, Feb. 1989.
3. Goodsell, Aga M.; Melton, John E.; and Madson, Michael D. "A Transonic/Supersonic CFD Analysis of a Generic Fighter." Presented at the 17th ICAS Congress 6.4.2., Sept. 1990.



Normalized stagnation pressure.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Simulation of Compressible, Spatially Evolving, Reactive Planar Shear Flows

Fernando F. Grinstein, Principal Investigator
Naval Research Laboratory

Research Objective

To understand the dynamics and topology of the large-scale coherent structures controlling the development of the mixing layers in transitional, reactive, free shear flows.

Approach

The approach is to solve numerically the three-dimensional, time-dependent, compressible-flow equations on structured grids using a fourth-order, phase-accurate flux-corrected transport (FCT) algorithm with directional time-step-splitting techniques, and appropriate inflow and outflow boundary conditions. No subgrid modeling other than that provided by the FCT high-frequency filter is included in the numerical model. Reactive flows are modeled using fast, one-step, irreversible chemical reactions.

Accomplishment Description

(1) The existence of global, self-sustaining instabilities in compressible (subsonic) mixing layers was demonstrated. (2) The effects of spanwise excitation, chemical reaction, and energy release on the development of free mixing layers were studied. (3) The three-dimensional coherent structures in the nonreactive plane jet and plane wake were investigated. The figure includes a typical instantaneous flow visualization from a numerical simulation of a nonreactive three-dimensional plane wake, showing the formation of unconnected, closed vortex loops in the high-strain saddle regions between rollers of opposite sign, in response to cross-stream spanwise excita-

tion. This particular 3000-step simulation of the plane wake was performed on the Cray Y-MP on a $170 \times 120 \times 60$ grid, using 8 megawords of storage and about 20 (single processor) CPU hours.

Significance

The simulations provide insight into the physics of large-scale coherent structures in planar shear flows and the mechanisms affecting the growth of mixing layers and the transition to turbulence. The effort advances the state of the art of transitional shear flow simulations, which may also be useful in developing the technology base for the National Aero-Space Plane.

Future Plans

Three-dimensional effects of compressibility and energy release will be investigated in simulations that have improved modeling of the chemistry and diffusive transport.

Publications

1. Grinstein, F. F.; Oran, E. S.; and Boris, J. P. "Reinitiation and Feedback in Global Instabilities of Subsonic Spatially Developing Mixing Layers." *Phys. Rev. Lett.* 64 (1990): 870.
2. Grinstein, F. F.; Hussain, F.; and Oran, E. S. "A Numerical Study of Mixing Control in Spatially Evolving Shear Flows." AIAA Paper 89-0977, 1989.
3. Grinstein, F. F.; Boris, J. P.; Griffin, O. M.; Hussain, F.; and Oran, E. S. "Coherent Structure Dynamics in Spatially Evolving Near Wake Flows." AIAA Paper 90-057, 1990.



Three-dimensionality in bluff-body wake; $U = 2 \times 10^2 \text{ m/s}$, grid size $170 \times 120 \times 60$.

Three-Dimensional Atmospheric Simulation Model

W. L. Grose, Principal Investigator

Co-investigators: W. T. Blackshear, R. W. Turner, and R. S. Eckman
NASA Langley Research Center

Research Objective

To conduct a three-dimensional simulation of the evolving atmospheric circulation and trace-constituent distributions over a multi-year cycle, in order to better understand stratospheric dynamics, transport, and photochemistry and to apply this knowledge in support of the Upper Atmospheric Research Satellite (UARS) program.

Approach

A three-dimensional, spectral, primitive-equation, atmospheric model incorporating chemistry has been developed. The model consists of two components: a module that calculates the winds and temperatures to describe the global atmospheric dynamics and a separate transport/chemistry module (driven by the simulated dynamics) to describe the transport of photochemically active trace constituents.

Accomplishment Description

A one-year simulation investigating aspects of the Antarctic ozone hole and its subsequent recovery following the breakup of the polar vortex was performed. Analysis of the results yielded information on the possible impact of the mixing of ozone-poor air into the midlatitudes in the southern hemisphere. In addition, a multi-year simulation to examine the

behavior of a long-lived tracer, nitrous oxide, has been run.

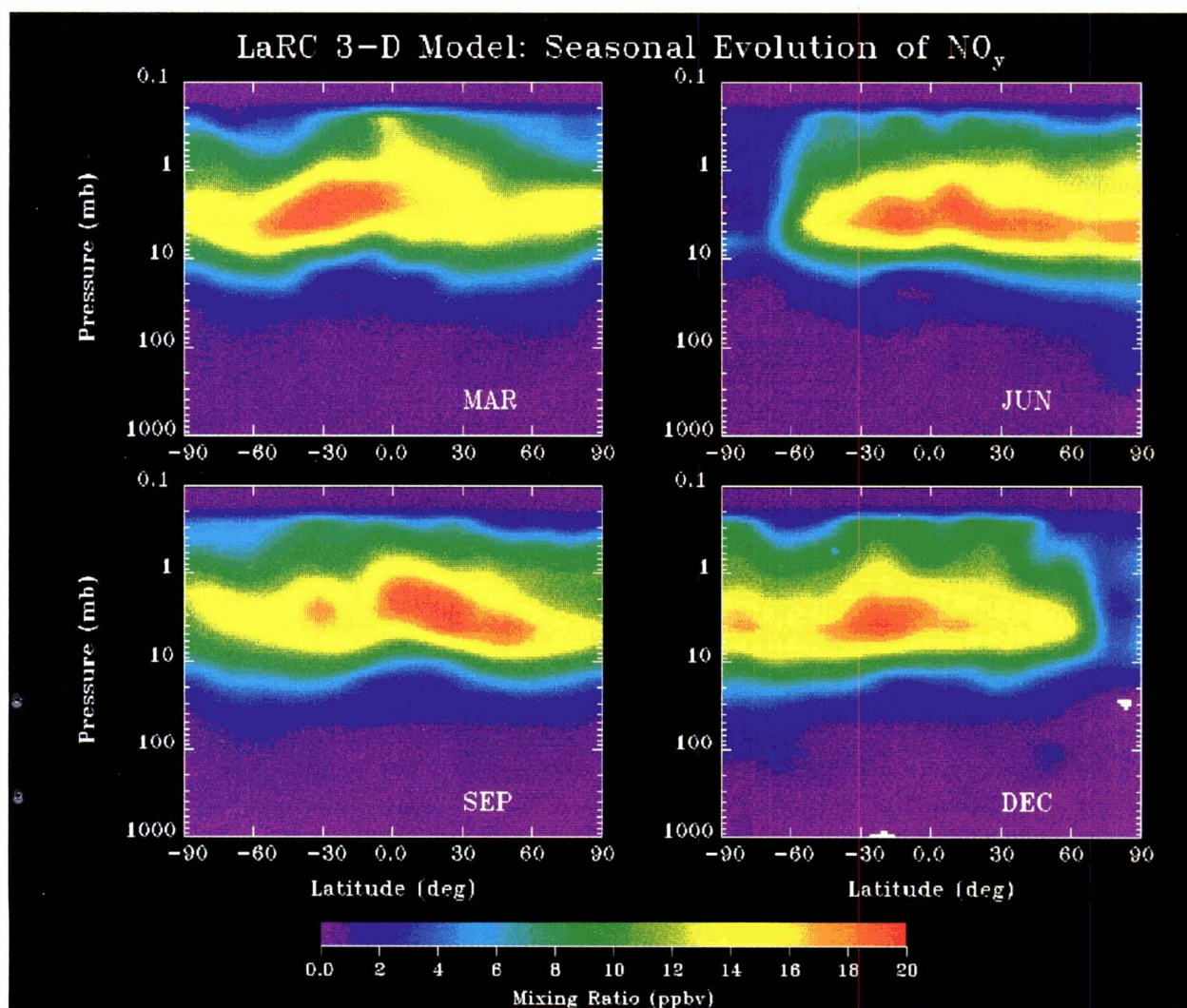
This simulation has enhanced our knowledge of aspects of stratospheric-tropospheric exchange processes. Analysis of a simulation to study the seasonal variation of the odd nitrogen family in the stratosphere is continuing. Typically, the model requires 300 CPU seconds per model day and 1.5 megawords of memory on the Cray Y-MP.

Significance

This study provides increased understanding of stratospheric dynamics, transport, and photochemistry, and supports the UARS and Earth Observing System (Eos) programs.

Future Plans

The transport/chemistry model is undergoing substantial revision to explicitly transport additional species, so that it will better represent high-latitude chemical processes. A parameterized heterogeneous chemistry scheme will be implemented to reflect our understanding of the importance of these processes in the stratosphere. A one-year simulation representing the Antarctic ozone depletion will be run to compare these model upgrades with our previous simulation.



Simulation of Fluid/Structural Interactions

G. P. Guruswamy, Principal Investigator

Co-investigators: S. Obayashi, N. M. Chaderjian, and P. M. Goorjian

NASA Ames Research Center

Research Objective

The objective is to develop a numerical capability to couple Euler/Navier-Stokes solutions with structural equations, for conducting aeroelastic analyses of complete aircraft. This capability is required for aircraft of national importance for defense, like the ATF, the F-16, and the B1 aircraft.

Approach

The three-dimensional Euler/Navier-Stokes equations of motion coupled with structural equations of motion are solved by finite-difference schemes.

Accomplishment Description

A code based on the Euler/Navier-Stokes equations, ENSAERO, is used to simulate unsteady flows over general wing configurations. The code uses implicit finite-difference methods for aerodynamic calculations based on both central and upwind schemes. The ability to directly couple the aerodynamic solutions from the Euler or Navier-Stokes equations with the structural equations for computing aeroelastic responses is incorporated in the code. The aeroelastic-configuration adaptive dynamic grids with zonal capability are used in the analysis. This is the first time that such a computational capability has been developed in computational fluid dynamics using the Navier-Stokes equations. Using ENSAERO, vortical and transonic flows are computed over flexible fighter wings in unsteady motions. A typical computation of a wing in ramp motion requires about 10 hours of CPU

time and 16 megawords of memory for a grid of size $151 \times 25 \times 32$.

Significance

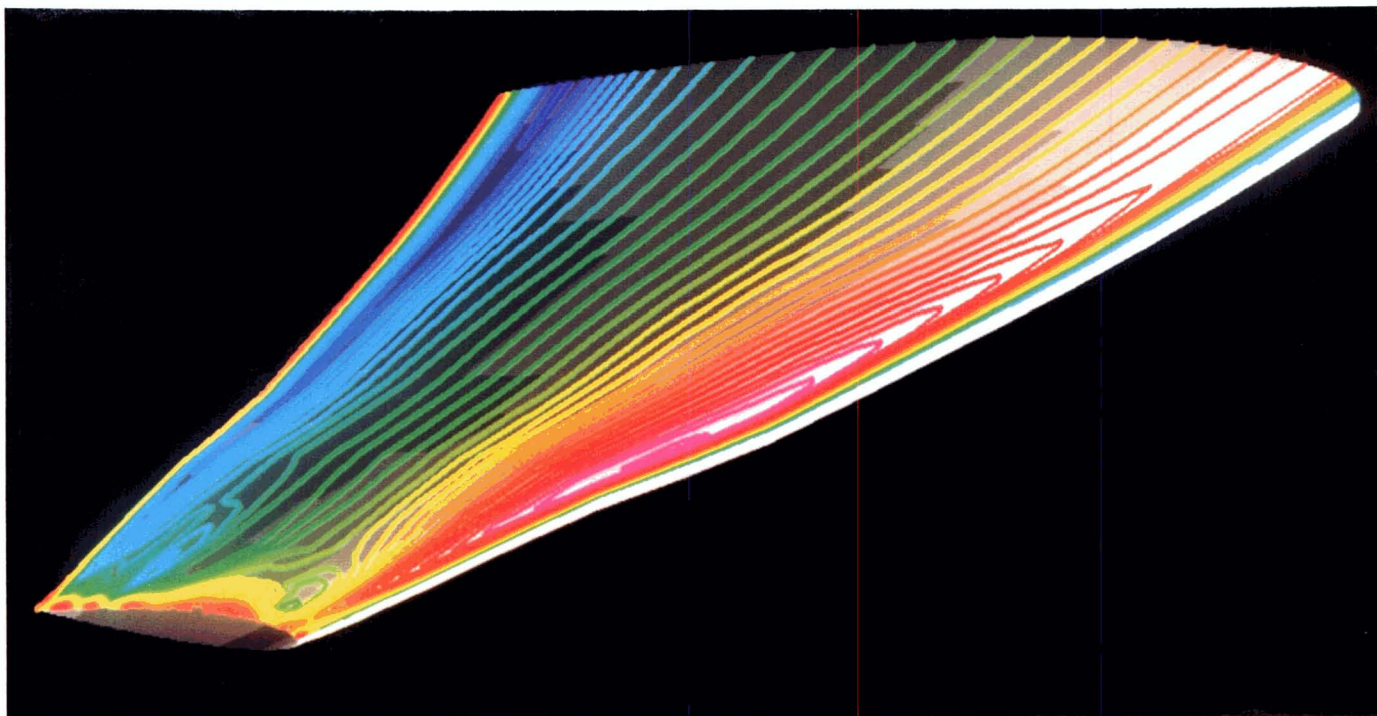
This project demonstrates the ability of computational methods to simulate aeroelasticity associated with complex vortical and transonic flows for improving the performance of an aircraft. It also serves as a major step toward developing a multidisciplinary version of ENSAERO for full-aircraft analysis. This development is of great national importance, particularly in view of the Air Force's ATF project.

Future Plans

Research will be continued to compute transonic and vortical flows over wing-body configurations and will be further extended to study aeroelastic responses associated with vortices and shock waves. In the coming years, ENSAERO will be developed as a multidisciplinary code to study fluid/structural interactions of aerospace vehicles, including thermal loads and controls.

Publications

1. Guruswamy, G. P. "Navier-Stokes Computations on Swept-Tapered Wings, Including Flexibility." AIAA Paper 90-1152-CP, Apr. 1990.
2. Chaderjian, N. M., and Guruswamy, G. P. "Unsteady Transonic Navier-Stokes Computations for an Oscillating Wing Using Single and Multiple Zones." AIAA Paper 90-0313, Jan. 1990.



Navier-Stokes computations of turbulent separated flows on a flexible wing in ramp motion; $AR = 4.0$, $TR = 0.3$, leading-edge sweep angle $= 30.0^\circ$, $M_\infty = 0.50$, $Re_c = 2.0 \times 10^6$, pitch rate $= 0.01$, $\alpha = 8.0^\circ$. Contours represent velocity magnitude. Flow is separated near the tip, leading to buffeting.

Vortex Dynamics in Aerodynamic Flows

Karl E. Gustafson, Principal Investigator
Co-investigator: Robert R. Leben
University of Colorado, Boulder

Research Objective

To validate large-scale vortex-separation patterns of accelerating, plunging, and pitching laboratory flows, and conversely to validate the codes simulating those flows, thereby making possible an increased understanding of the aerodynamic characteristics of such flight.

Approach

We use a two-dimensional stream-function vorticity scheme that includes a novel grid-generation scheme in which the infinite physical domain is mapped to auxiliary flow-computation domains, avoiding any truncated far-field boundary conditions. An orthogonal grid using a weak interpolation constraint is obtained by an efficient multigrid solver, which is used as well in the unsteady Navier-Stokes solver.

Accomplishment Description

Laboratory findings of vortex-splitting and vortex-shredding events in acceleration flows over airfoils were validated and, in most cases, reproduced exactly by our code flow simulations. Significant lift increase caused by such events near dynamic stall was exhibited. More recently we have successfully modeled several complex combined plunging/pitching modes

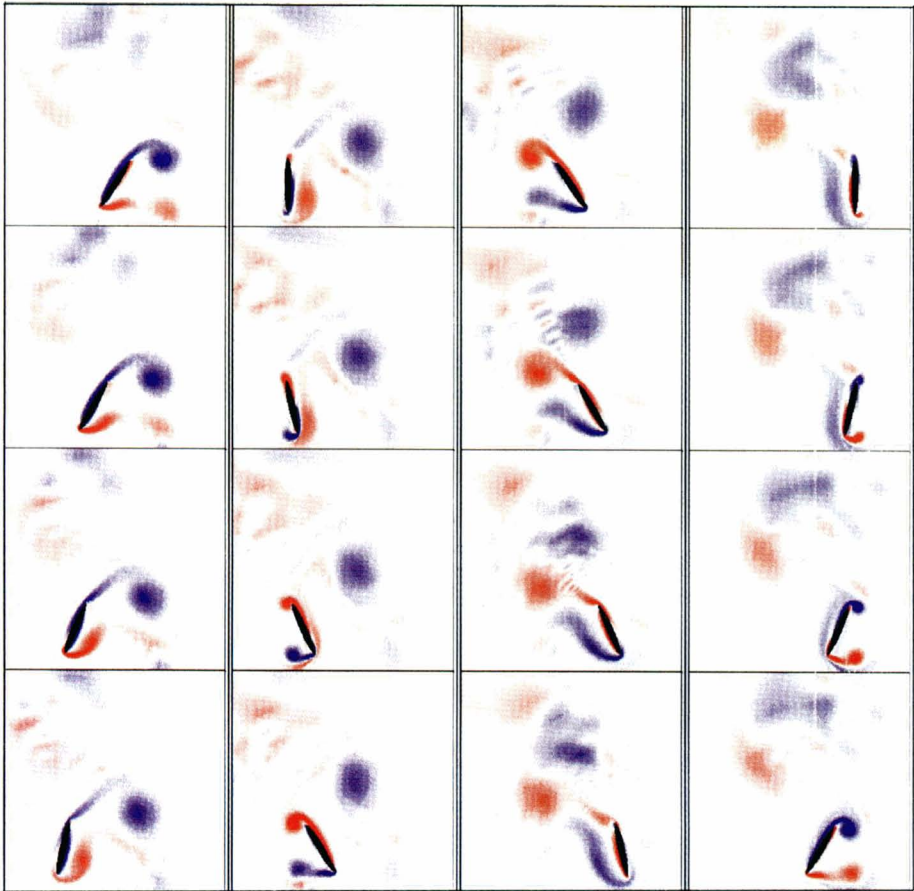
representing hovering flight in still air. The result is a self-induced large-thrust jet formed by matched vortical patterns exhibiting rotations reversed from those of a Karman street. The figure shows a segment of a period of such a hovering mode, in which laboratory visualizations indicated a vortex-severing event, seen clearly in the numerical simulation. A typical computation on a 65×65 grid required 10 megawords of Cray-2 memory and approximately 40 minutes of CPU time for four periods of a typical hovering-mode computation, including graphics processing.

Significance

Hovering-mode flight in still air, as practiced, for example, by dragonflies and other insects and small birds, exhibits the construction of dynamic stall vortices that enable the production of very high thrust. The simulation of such modes will make possible a considerably increased understanding of the unsteady aerodynamic characteristics (e.g., lift) of such flight.

Future Plans

The main immediate focus will be to carefully calculate the aerodynamic lift to compare with the downstream thrust coefficient found in the laboratory studies.



A vortex-severing event for an aircraft in hovering mode.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Parameter Study in the Driven Cavity

Kadosa Halasi, Principal Investigator

Co-investigator: Karl E. Gustafson

Kansas State University/University of Colorado, Boulder

Research Objective

To investigate the effects of parameters on two-dimensional incompressible flow. In particular, to determine critical parameter values at which asymptotic solutions become periodic.

Approach

Two-dimensional, time-dependent, incompressible Navier-Stokes equations expressed in the biharmonic stream-function form are used. The convection terms are differenced with an Adams-Bashford scheme, and a Crank-Nicolson method is applied to the diffusion terms. A multigrid solver is used to solve the linear implicit equation at each time step.

Accomplishment Description

For an aspect ratio of $A = \text{width/depth} = 2$, the critical Hopf bifurcation value of Reynolds number (Re) was sought. Measures of periodicity were introduced, and the periodicity (or lack of periodicity) of the flow with respect to these measures was followed. The effects of grid-mesh size on periodicity was also investigated. It was found that (1) the critical Re satisfied $2000 < Re < 5000$; (2) the time to achieve a periodic state increased with decreasing mesh size, and the period, itself,

decreased on finer meshes; and (3) there existed a period of intermittency, during which time the flow was caught between a tendency toward a steady final state on one hand, and a periodic final state on the other.

Significance

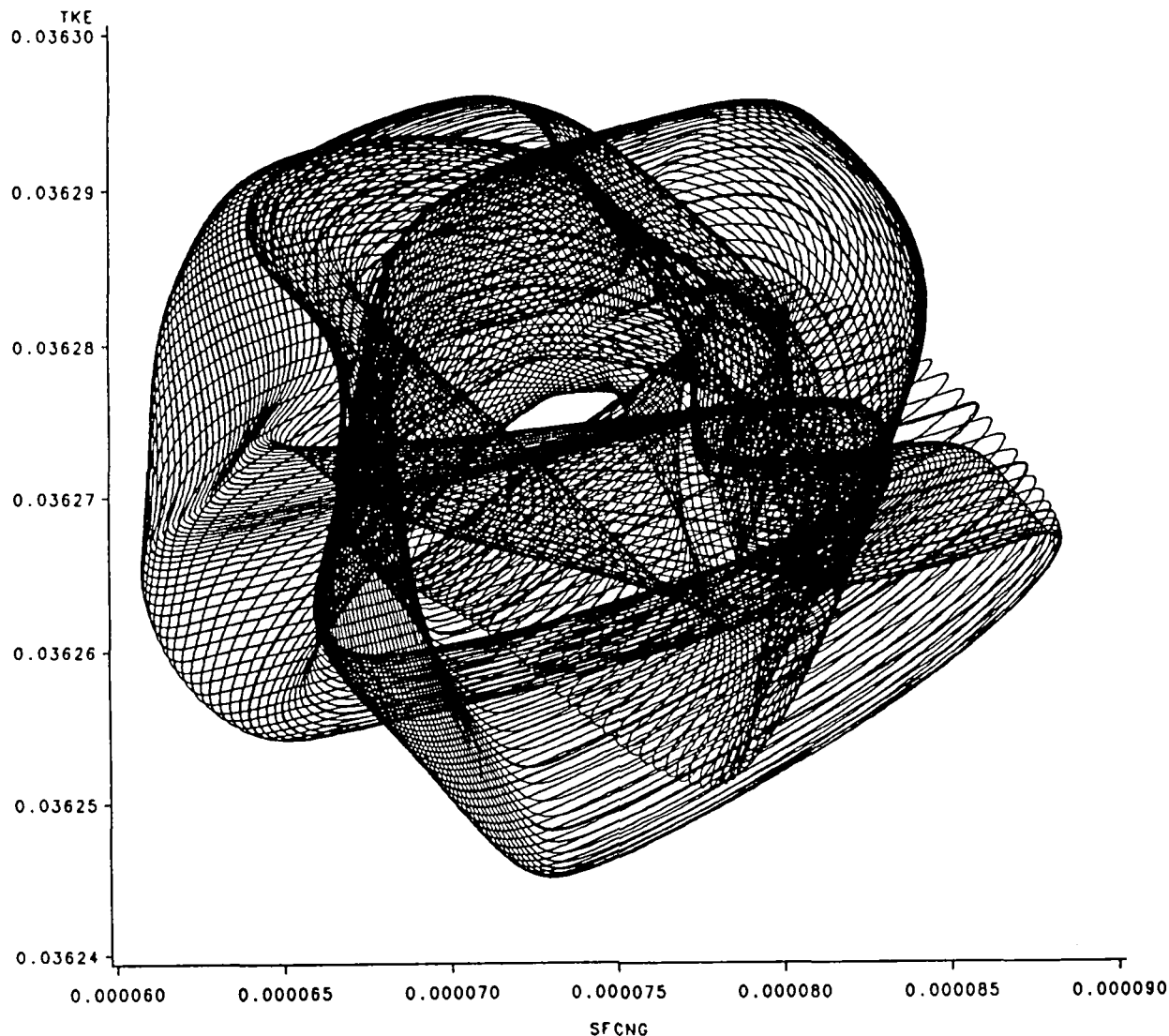
The investigation will provide a better understanding of vortex dynamics in the face of an impending oscillatory flow regime.

Future Plans

We plan to continue the parameter study in both shallow and deep cavities, and to extend the study to three-dimensional settings.

Publications

1. Goodrich, J.; Gustafson, K.; and Halasi, K. To be published in *J. Computational Physics*, 1990.
2. Halasi, K.; Gustafson, K.; and Goodrich, J. "Periodicities, Intermittencies, and Criticalities in Driven Cavity and General Flows." Presented at IMACS 1st International Conference on Computational Physics, Boulder, CO, June 1990.



Stream-function change versus total kinetic energy, $6100 < t \leq 7100$.

Ducted Propfan Analysis

Edward J. Hall, Principal Investigator
Allison Gas Turbine Division, General Motors Corporation

Research Objective

The ultimate objective of this research is to develop an advanced aerodynamic analysis tool for ducted-fan aircraft propulsion systems. This analysis will provide a detailed description of the flow through the fan and the external flow about the fan cowl and nacelle. Special attention is given to the accurate prediction of the flow near the fan-cowl leading edge.

Approach

A three-dimensional, time-marching, finite-volume Euler analysis is applied to predict the steady flow about ducted propfan geometries. The Euler analysis is based on a multiple-grid-block solution scheme that uses a cylindrical coordinate system that rotates with the fan. The multiple-grid-block solution technique permits a number of convenient grid arrangements, to enhance the accuracy of the calculated results.

Accomplishment Description

An explicit Runge-Kutta Euler code was modified to utilize a multiple grid-block strategy for the prediction of high-speed ducted-fan flows. Calculations were performed on both an H-type grid system incorporating a branch-cut-opening representation for the cowl, and a multiple-block structured grid using a C-type mesh about the cowl leading edge. The multiple grid blocks were numerically coupled through the mutual specification of boundary conditions along the interface between adjacent blocks. The H-grid analysis was extended to permit the prediction of counter-rotating flows through the use of the average-passage solution technique. Both single-rotation and counter-rotation calculations were performed. Predicted results were compared with numerical

and experimental data for several ducted-fan geometries. A typical solution required 1 CPU hour and 2 megawords of memory.

Significance

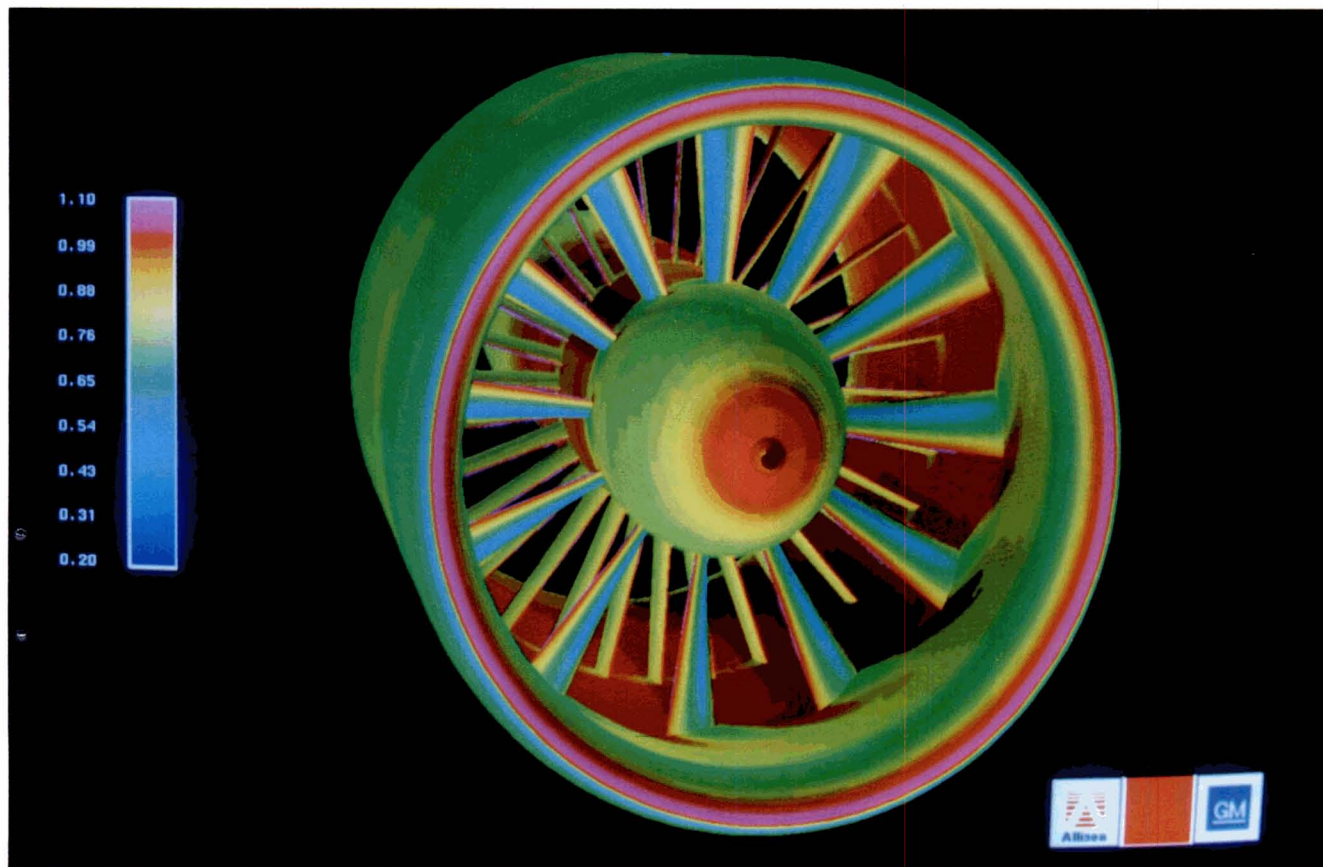
The principal benefit realized by the very high bypass ratio (VHBPR) turbofan engine is a substantial improvement in propulsive efficiency over current high-bypass-ratio engines. However, VHBPR engines impose stringent demands on the aerodynamic and structural integration of large, variable-pitch airfoils in minimum-length, low-drag cowls. This program is the initial step in developing a complete aerodynamic model of the flow field through a ducted fan and the external flow over the engine nacelle and fan cowl.

Future Plans

The multiple-block Euler analysis will be modified to include a Navier-Stokes solution capability. Following the incorporation of time-accurate boundary conditions, the time-marching scheme will be applied to predict the unsteady flow resulting from ducted propfans operating at angle of attack.

Publications

1. Hall, Edward J.; Delaney, Robert A.; and Bettner, James L. "Investigation of Advanced Counterrotation Blade Configuration Concepts for High Speed Turboprop Systems: Task 1 - Ducted Propfan Analysis." NASA CR-185217, Apr. 1990.
2. Hall, Edward J.; and Delaney, Robert A. "3D Euler Analysis of Ducted Propfan Flowfields." Presented at the AIAA Applied Aerodynamics Conference, Portland, OR, Aug. 1990.



Predicted static pressure ratio for an Allison ducted propfan; $M = 0.75$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Evaluation of the CAP-TSD Computer Code Using an F-16 Study Case

Richard L. Haller, Principal Investigator
Co-investigators: Kevin Penning and Lam-son Vinh
General Dynamics, Fort Worth Division

Research Objective

To determine if the computer code CAP-TSD can predict limit cycle oscillations for an F-16.

Approach

The unsteady aerodynamics code CAP-TSD will be used to model the F-16 aircraft. The CAP-TSD results will be compared with available test data. The CAP-TSD code will then be used to analyze a store loading that experiences limit cycle oscillations.

Accomplishment Description

Steady-state CAP-TSD results have been compared with wind tunnel test data for Mach numbers 0.85, 0.95, and 1.2 at angles of attack of 2° and 4°. The comparisons were good and the work is continuing into unsteady aerodynamics. Oscillatory pressures are not available for the F-16 so comparisons will be

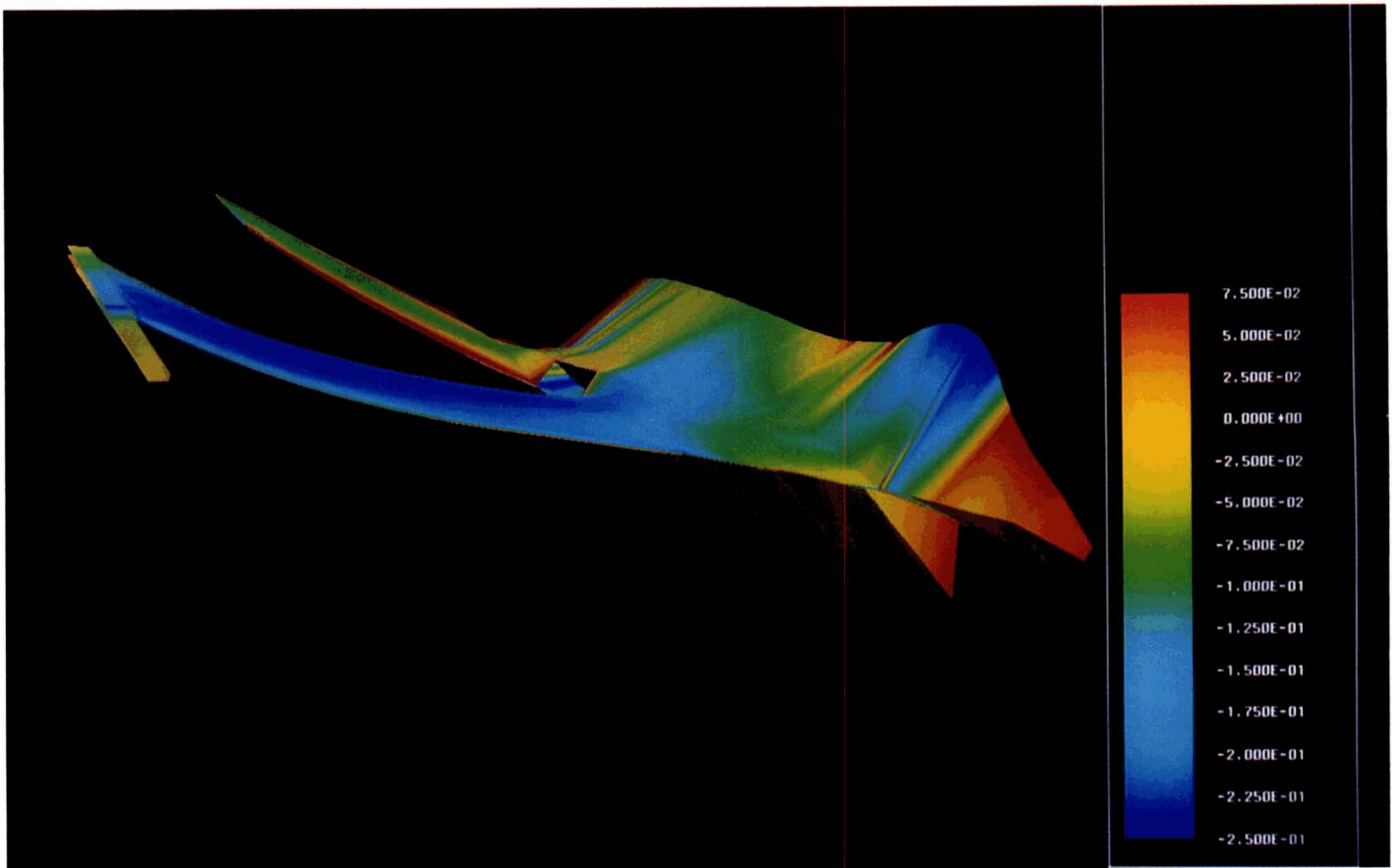
made between the predicted and measured flutter speeds. Because of our lack of experience with the CAP-TSD code and the long turnaround time on the NAS computers, only 10 hours of Cray-2 time and 32 megawords of memory were used.

Significance

The CAP-TSD code is presently the only computer code available to industry to analyze the unsteady aerodynamics of three-dimensional aircraft in transonic flight. It represents a large step forward from the doublet-lattice and kernel-function codes that are in use today. In order to bring this code into use in industry, CAP-TSD must be evaluated and guidelines for its use developed.

Future Plans

The CAP-TSD code will be used to analyze a store loading that experiences limit cycle oscillation.



The CAP-TSD upper surface pressure coefficients and deflection for an F-16 aircraft at $M = 0.8$ and $\alpha = 2^\circ$. (Deflections magnified by a factor of 10.)

Supersonic Laminar-Flow-Control Computational Fluid Dynamics

Julius E. Harris, Principal Investigator
Co-investigators: Venkit Iyer and Robert E. Spall
NASA Langley Research Center

Research Objective

To develop numerics and related software for three-dimensional transition prediction (TP) and laminar flow control (LFC) for high-speed aircraft.

Approach

Two three-dimensional compressible boundary layer (3DBL) codes have been developed and are being verified. The general TP/LFC software package includes (1) the 3DBL packages, (2) two Euler solvers (ES), (3) grid generations for the 3DBL and ES packages, and (4) an interface/output package for the existing stability/transition routines (COSAL, EMALIK). Three-dimensional Navier-Stokes solutions are also required in order to verify the TP/LFC software and determine problems associated with non-boundary layer equation regions (wing-body, for example). Current emphasis is being placed on obtaining Euler and Navier-Stokes solutions for the F16xL configuration. The pressure field from the Euler solutions will be used as input for the 3DBL software. The output from the Navier-Stokes solution will be used in the validation of the Euler plus 3DBL solutions. Mach number contours using the CFL3DE Euler solver contours for the upper surface of the

F16xL wing at an angle of attack of 4° for a free-stream Mach number of 2 is presented in the figure. Typical system requirements per converged case are as follows: 3DBL (0.5 megawords of memory, 10 minutes CPU time), Euler (8 megawords of memory per block, 6 blocks, 4 hours CPU time), Navier-Stokes (16 megawords of memory per block, 6 blocks, 40 hours CPU time).

Significance

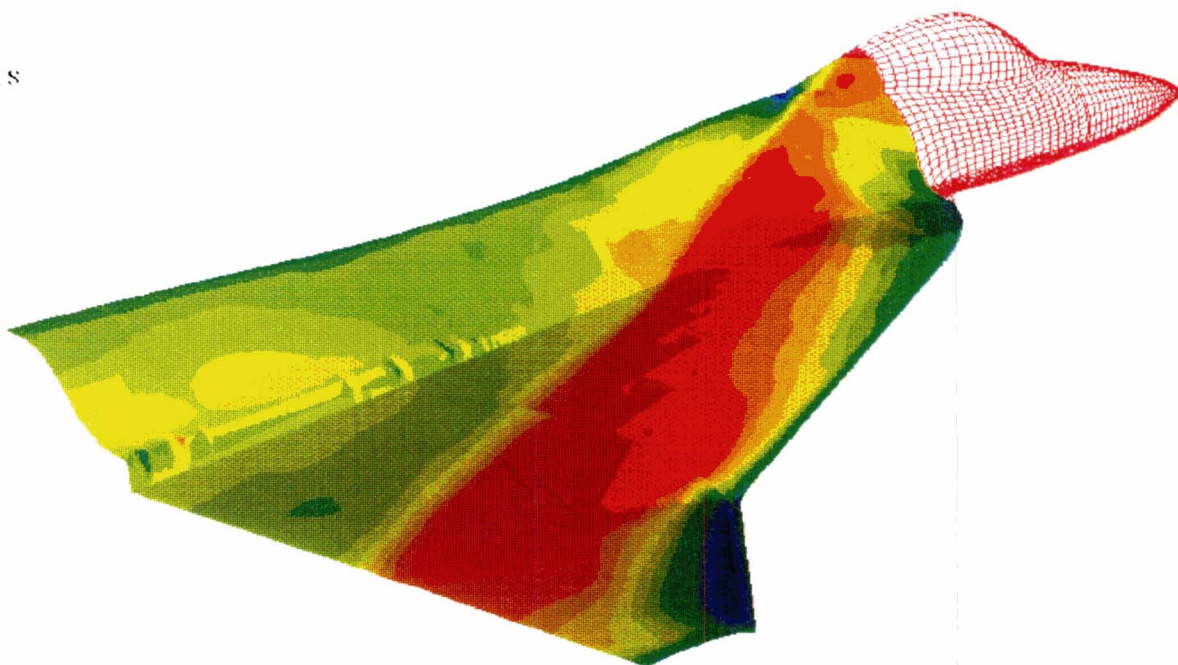
This work has a direct impact on high-speed LFC and transition prediction and control.

Publications

1. Iyer, V., and Harris, J. E. "Fourth-Order Accurate Three-Dimensional Compressible Boundary-Layer Calculations." *J. Aircraft* 27, no. 3 (Mar. 1990): 253-261.
2. Wie, Yong-Sun, and Harris, J. E. "Numerical Solution of the Boundary-Layer Equations for a General Aviation Fuselage." AIAA Paper 90-0305, 1990.
3. Iyer, Venkit. "Computation of Three-Dimensional Compressible Boundary Layers to Fourth-Order Accuracy on Wings and Fuselages." NASA CR-4269, 1990.

CONTOUR LEVELS

1.50000
1.55000
1.60000
1.65000
1.70000
1.75000
1.80000
1.85000
1.90000
1.95000
2.00000
2.05000
2.10000
2.15000
2.20000
2.25000
2.30000
2.35000
2.40000
2.45000
2.50000



Mach number contours for the F16xL wing upper surface, from the CFL3DE Euler solver; $M = 2.0$, $\alpha = 4^\circ$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

General, Fast, Accurate Floating Shock Fitting Procedure for Unadapted Meshes

Peter M. Hartwich, Principal Investigator
ViGYAN, Inc./NASA Langley Research Center

Research Objective

To develop an accurate and efficient method for computing shocked flows on relatively crude, unadapted meshes.

Approach

A time-implicit upwind Euler solver is constructed. Code development is expedited by relying as much as possible on proven numerical techniques and by initially focusing on two-dimensional flows. For computational speed, the upwinding is based on the nonconservative split-coefficient-matrix method, and for accuracy across shocks, floating shock fitting is implemented.

Accomplishment Description

Results have been obtained for Riemann's problem, for a regular shock reflection at an inviscid wall, for supersonic flow past a cylinder, and for a transonic airfoil. In all these cases, the computed shocks are in good to excellent agreement with reference solutions. This is exemplified by the results shown in the accompanying figure for supercritical flow ($M_\infty = 0.8$) over a NACA 0012 airfoil at zero incidence. These solutions were computed on a coarse and a standard C-type mesh. The standard mesh with 161×33 grid points is closely patterned after that used by Pulliam et al. (AIAA Paper 83-0344) for subcritical flow; the coarser grid is obtained by dropping every other grid point in each coordinate direction. For the chosen parameters, shocks develop at $x/c \approx 0.5$, which coincides with the coarsest chordwise resolution of the "subcritical" grids. The spatial step size $\Delta x/c$ in the neighborhood of the fitted shocks is about twice (161×33 grid) and four times (81×17 grid) as large as in the reference solution, which was computed on an adapted 161×33 grid with sixteen radial lines clustered along the shock fronts with a regular spacing of $\Delta x/c = 0.01$. Even with that handicap, the surface pressure distributions and the shock locations as predicted by the present method on both grids agree well with the accurate shock-capturing results on the adapted grid. (Note that despite their crisp appearance, the captured shocks are still resolved over four mesh intervals.) The surface entropy distri-

butions provide another means of demonstrating the accuracy of the present results: (1) entropy is only generated across the shocks, (2) away from the shocks entropy is merely convected and thus it remains at a constant level, and (3) the fitted shocks are completely devoid of any spurious oscillations. Starting from free-stream conditions, it took about 500 to 600 iterations to reduce the L_2 -norm of all residuals by at least four orders of magnitude and to establish asymptotic values for the drag, which differ by just four counts for the two cases. Computing these results on a single processor of the NAS Cray Y-MP required about $6.5 \mu\text{sec}/\text{node}/\text{iteration}$, which translates to total run times of 4.5 sec-onds for the 81×17 grid and roughly 18 seconds for the 161×33 grid.

Significance

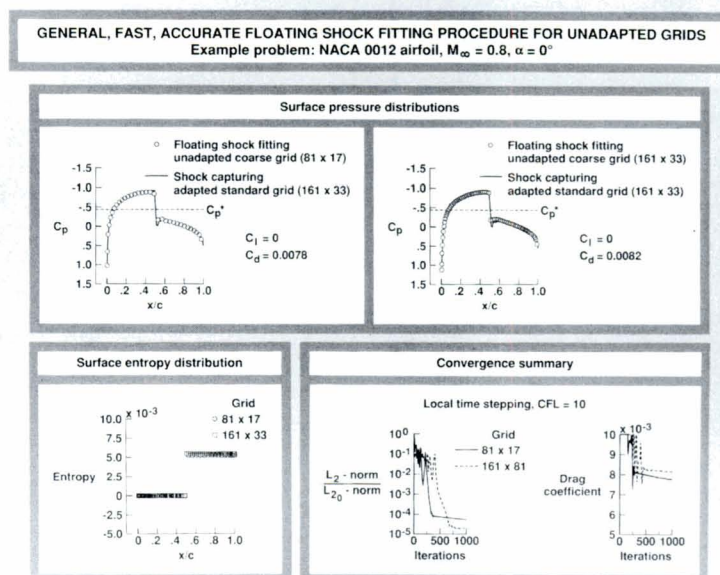
The present general shock-fitting procedure for solving the two-dimensional Euler equations at least matches, if not surpasses, the computational efficiency of shock-capturing schemes. Even more importantly, it produces highly accurate solutions for shocked flows which exhibit very little grid sensitivity. This brings down costs in two ways: savings in the total number of grid points required for a proper resolution of the flow translate to immediate shorter computing times, and reducing the need to generate adapted meshes improves turnaround time.

Future Plans

The present floating shock fitting technique is currently being extended to three dimensions. Further extensions will encompass the addition of viscous terms, a turbulence model, and a capability to handle overset grids.

Publications

1. Hartwich, P. M. "Fresh Look at Floating Shock Fitting." AIAA Paper 90-108, Jan. 1990.
2. Hartwich, P. M. "Split Coefficient Matrix (SCM) Method with Floating Shock Fitting for Transonic Airfoils." Presented at the 12th International Conference on Numerical Methods in Fluid Dynamics, Oxford, UK, July 1990.



Supercritical flow over a NACA 0012 airfoil; $M = 0.8$, $\alpha = 0^\circ$. (Top) Surface pressure distributions. (Bottom left) Surface entropy distribution. (Bottom right) Convergence summary.

Navier-Stokes Solutions for Vortical Flows over a Forebody

Peter M. Hartwich, Principal Investigator
ViGYAN, Inc./NASA Langley Research Center

Research Objective

The research objective is the computational assessment of the effects of Reynolds number and angle of attack on vortical flows over a pointed slender body.

Approach

Steady-state solutions for vortical flows with $0.2 \times 10^6 \leq Re_D \leq 3.0 \times 10^6$ and $20^\circ \leq \alpha \leq 40^\circ$ over a 3.5-caliber tangent-ogive cylinder are computed with FMC1, a time-implicit upwind method for the three-dimensional, incompressible Navier-Stokes equations. This solver comprises flux-difference splitting, a TVD-like discretization of the inviscid fluxes, and several extensions of the algebraic turbulence model by Baldwin and Lomax to handle flows with massive crossflow separation.

Accomplishment Description

A rational extension of the Baldwin-Lomax turbulence model has been devised. That extension allows computational modeling of transitional crossflow separation (i.e., flows with three-dimensional, laminar, equatorial separation bubbles and subsequent transition in the separating shear layers which roll up into two primary vortices). In addition, it proves to be relatively insensitive to the choice of adjustable parameters in simulations of fully turbulent crossflow separations. For $\alpha \geq 30^\circ$ and $Re_D = 0.8 \times 10^6$, a perturbation of the geometry into a slightly elliptic cross section just at the nose tip eliminates multiple steady-state solutions with asymmetric vortex patterns along the forebody. A typical solution is shown in the accompanying figure, in which the major axis of an "elliptic" nose tip is rolled by 45° in the counterclockwise direction (pilot's view) out of its horizontal position. Four shedding events are shown by means of helicity density contours (helicity density is

defined as the scalar product of local velocity and vorticity vectors). An almost perfect mirror image is produced when the nose tip is rolled for another 90° . This supports earlier conjectures that slight imperfections in the vicinity of the apex of slender bodies with sharp noses control the asymmetric pattern along the entire body. Reynolds number effects for $\alpha \leq 30^\circ$ are found to diminish for $Re_D \geq 1.0 \times 10^6$. All computational results compare well with experimental surface pressures and flow visualizations. Typical calculations of symmetric forebody flow are carried out on grids of about 190,000 grid points, and they require about 1.5 Cray-2 hours (single processor mode). A fully converged result for asymmetric vortical flow on a standard grid of about 380,000 grid points is obtained after about 2500 time steps, which translates to roughly 10 Cray Y-MP hours (single processor mode).

Significance

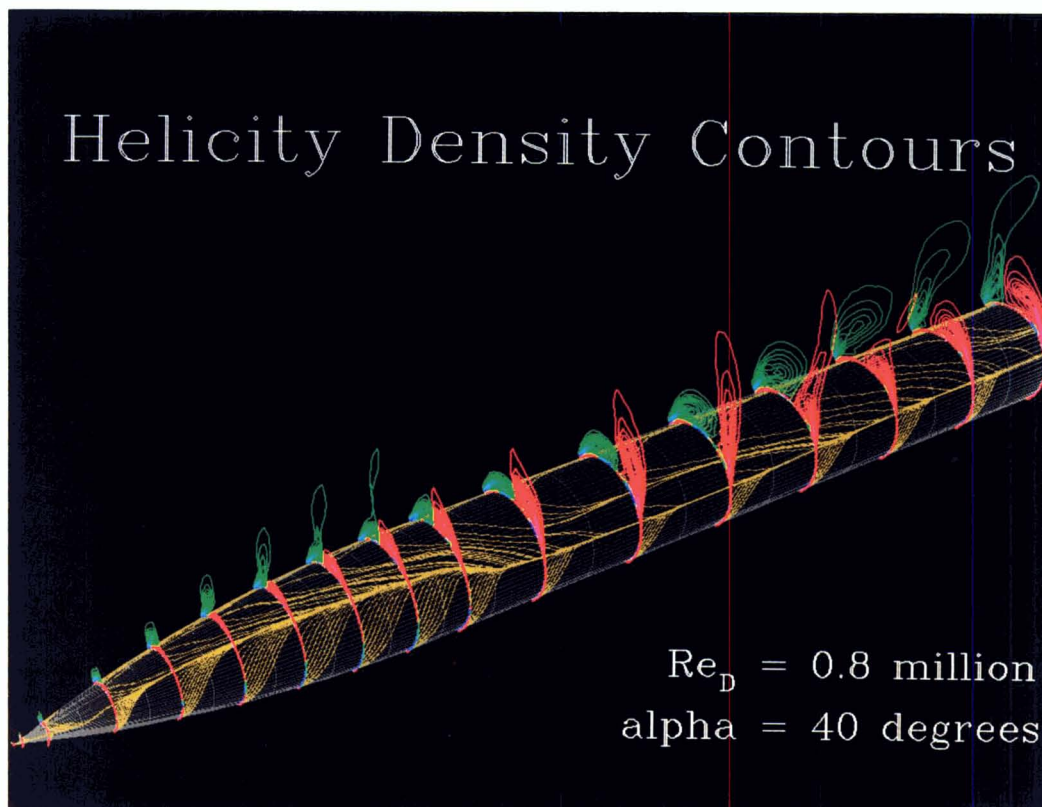
Knowledge about transition and other driving mechanisms controlling vortex asymmetries will assist in the design of improved missiles and fighter aircraft forebodies.

Future Plans

A Navier-Stokes analysis of an F/A-18 or an F-16 forebody with a modified nose tip will exploit the methodology and the insights into the physics of vortical forebody flows.

Publications

1. Hartwich, P. M.; Hall, R. M.; and Hemsch, M. J. "Navier-Stokes Computations of Vortex Asymmetries Controlled by Small Surface Imperfections." AIAA Paper 90-0385, Jan. 1990.
2. Hartwich, P. M., and Hall, R. M. "Navier-Stokes Solutions for Vortical Flows Over a Tangent-Ogive Cylinder." AIAA J. 28 (July 1990).



Helicity density contours; $\alpha = 40^\circ$, $Re_D = 0.8 \times 10^6$.

Monte Carlo Simulation of Hypersonic Flows

H. A. Hassan, Principal Investigator

Co-investigators: David P. Olynick and Jeff C. Taylor
North Carolina State University

Research Objective

The objective of this work is to calculate nonequilibrium radiation for reentry vehicles. The heat load resulting from such radiation becomes important for reentry velocities in excess of 10 km/sec.

Approach

Solutions of the Navier-Stokes equations for reentry flows break down when the local Knudsen number is 0.2 or higher. Moreover, when thermal nonequilibrium is important, the functional dependence of reaction rate on the various temperatures is not known, thus rendering Navier-Stokes solutions useless. As a result, an approach based on the direct simulation Monte Carlo (DSMC) method is used.

Accomplishment Description

All DSMC methods depend to a large extent on the physical model used. Traditional DSMC computations have a number of limitations. Internal energy partition relies on a phenomenological model, and probabilities that determine energy partition are assumed constant. Moreover, available dissociation rates seem to overpredict dissociation at high temperatures. Finally, when ionization is important, a consistent treatment of ambipolar diffusion is needed. All of these issues have to be resolved before nonequilibrium radiation can be cothe actual distribution for various times. a heating environment from 4,000

to 8,000 K was undertaken. The accompanying figure shows that during the vibrational relaxation of rotationless N_2 from 4,000 to 8,000 K, all levels remain in equilibrium at the average vibrational temperature. Because of this, the bottleneck observed in continuum calculations is not observed here. A typical job requires 4 megawords of memory and 40 CPU hours of run time.

Significance

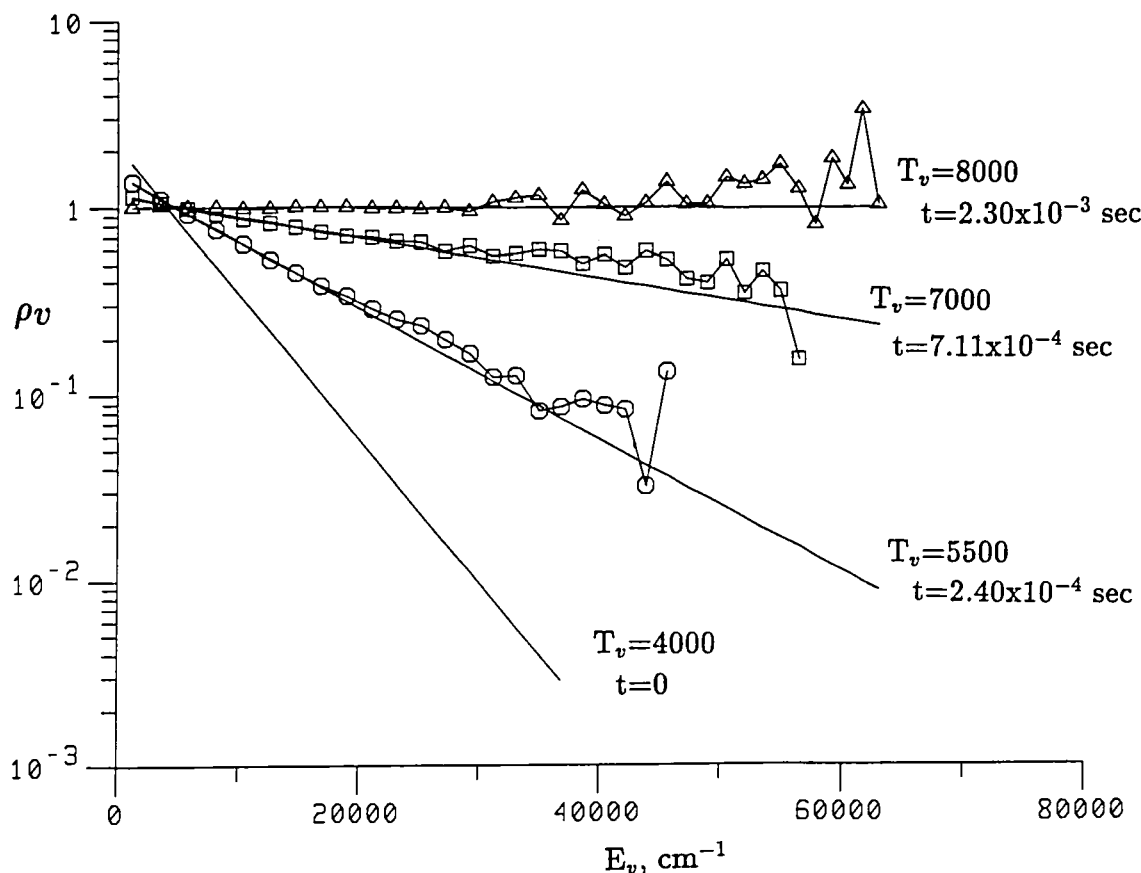
Proposed missions to Mars and return require calculation of nonequilibrium flows that may not be modeled correctly by the Navier-Stokes equations. Therefore, it is important that an alternative simulation procedure be developed and perfected.

Future Plans

We plan to continue improving the physical models used in the DSMC. Moreover, there is always a need to speed up the simulation.

Publications

1. Olynick, David P.; Moss, James N.; and Hassan, H. A. "Influence of Afterbodies on AOTV Flows." AIAA Paper 89-0311, Jan. 1989.
2. Taylor, J. C.; Moss, J. N.; and Hassan, H. A. "Study of Hypersonic Flow Past Sharp Cones." AIAA Paper 89-1713, June 1989.



Comparison of the equilibrium vibrational population distribution at the average vibrational temperature with the actual distribution for various times.

Turbulence Modeling of Separated Flows

H. A. Hassan, Principal Investigator

Co-investigators: Robert A. Mitcheltree and Richard L. Gaffney, Jr.
North Carolina State University

Research Objective

To develop turbulence models capable of predicting attached and separated flows for a wide range of Mach numbers.

Approach

We are developing a Navier-Stokes code that incorporates a turbulent stress model. The development is being carried out using airfoil data at various Mach numbers and angles of attack. The current model uses field equations for the shear stress and the turbulent kinetic-energy equation, and algebraic expressions for the normal stresses and the turbulent energy-dissipation rate.

Accomplishment Description

An explicit upwind code with multigrid, based on Roe's method, was developed. The time stepping was based on a four-step Runge-Kutta method. The constants in the turbulent model were fine-tuned for a flat plate and are being used for flow past airfoils. Earlier work used a one-equation eddy viscosity model and a separation model that incorporated observations made by Simpson. Unfortunately, it was not able to match the success of the Johnson-King model. More recently, a new stress was incorporated. The model is still under development. The figure shows the prediction of the stress model, and other models, of the pressure distribution for case 9 of the RAE 2822 airfoil. A job size is about 4 mega-words and requires 2 CPU hours to converge.

Significance

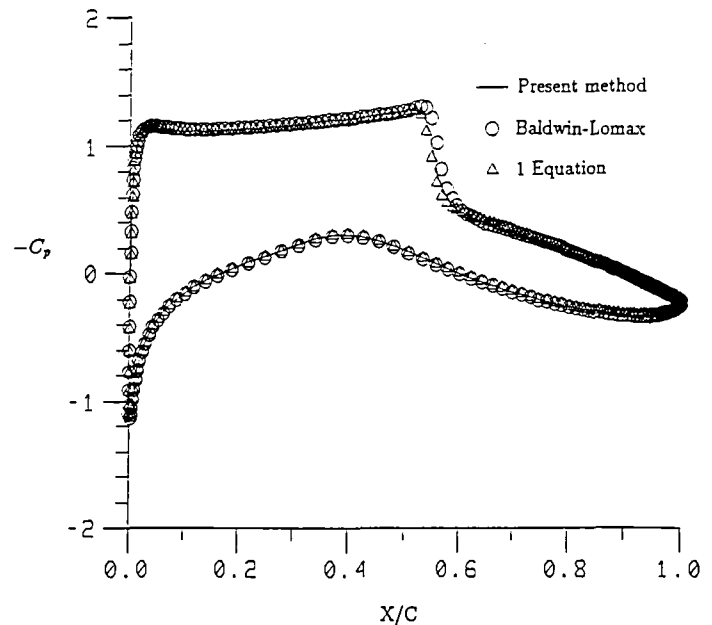
Designing high-performance, high-angle-of-attack aircraft requires the understanding of turbulent separated flows. Without such understanding, it is not possible to determine with confidence the aircraft's performance and characteristics.

Future Plans

Some of the assumptions made regarding the normal stresses and the rate of energy dissipation will be relaxed. An equation for the inverse of the turbulent time scale will be implemented in place of an equation for energy dissipation, because it appears to be more appropriate.

Publications

1. Mitcheltree, R. A.; Salas, M. D.; and Hassan, H. A. "A One Equation Turbulence Model for Transonic Airfoil Flows." AIAA Paper 89-0557, Jan. 1989.
2. Gaffney, R. L., Jr.; Salas, M. D.; and Hassan, H. A. "An Abbreviated Reynolds Stress Turbulence Model for Airfoil Flows." AIAA Paper 90-1468, June 1990.



Pressure distribution predictions for case 9 of the RAE 2822 airfoil.

Turbulent Supersonic Mixing Layers

H. A. Hassan, Principal Investigator

Co-investigators: Dean R. Eklund, Steven H. Frankel, and Erick J. S. Gantt
North Carolina State University

Research Objective

The ultimate goal of this work is to develop the computational capability to predict turbulent supersonic combustion in scramjets. The early phase of this work is restricted to mixing layers that simulate parallel injection of hydrogen fuel into a scramjet combustor.

Approach

An axisymmetric Reynolds-averaged Navier-Stokes code with nonequilibrium chemistry was developed and used to predict the flow field in a coaxial jet. Early work treated the source terms in the species conservation equations as laminar-like; more recent work used an assumed probability density function to calculate the averages of the source terms. The results were used to predict the results of two experiments involving supersonic mixing layers.

Accomplishment Description

Early work used an algebraic turbulence model to model coaxial mixing layers in the absence of combustion. The constants in the turbulence model were fine-tuned using experiments, carried out by Eggers, involving coaxial hydrogen-air jets. Next, two experiments involving combustion, one by Beach and the other by Jarrett et al., were modeled. It became clear that the constants developed for cold flows were not

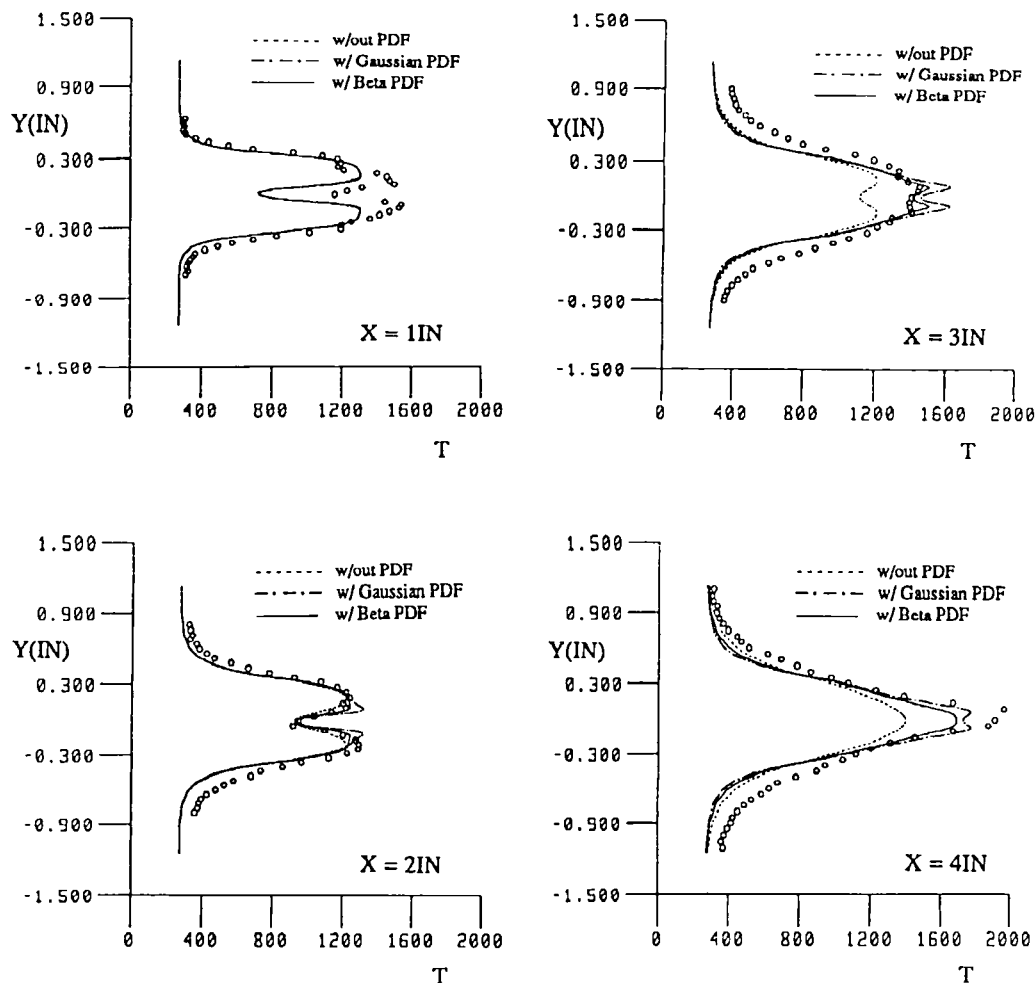
suited for modeling flows with combustion. This confirms the importance of the coupling of mixing and combustion. The source terms in the calculations were evaluated at the mean flow properties and, as such, were treated as laminar-like. More recent work used two assumed probability density functions; one was Gaussian and the other was a beta function. The figure compares some of the results with the measurements of Jarrett. A job size is about 4 mega-words and requires 2 CPU hours to converge.

Significance

The success of the National Aero-Space Plane and similar vehicles will depend on the availability of an efficient scramjet engine. Because of the lack of suitable test facilities, the development of the engine will have to rely on adequate physical and computer models.

Publications

1. Eklund, D. R.; Drummond, J. P.; and Hassan, H. A. "Numerical Modeling of Turbulent Supersonic Reacting Coaxial Jets." AIAA Paper 89-0660, Jan. 1989.
2. Frankel, S. H.; Drummond, J. P.; and Hassan, H. A. "A Hybrid Reynolds Averaged/PDF Closure Model for Supersonic Turbulent Combustion." AIAA Paper 90-1593, June 1990.



The flow field in a coaxial jet; the results of computations with and without an assumed probability density function (PDF) are compared with experimental results.

Multiblock Solutions of the Navier-Stokes Equations

Henry J. Haussling, Principal Investigator

Co-investigators: Joseph J. Gorski and Roderick M. Coleman

David Taylor Research Center

Research Objective

To apply and validate numerical methods for a multiblock solution of the Reynolds-averaged Navier-Stokes equations for turbulent flows about naval vehicles.

Approach

The multiblock capability of the DTNS3D method and computer code is being developed for and tested on the computation of flows about submerged bodies with and without appendages. The grid-generation capability of the computer code NUGGET is being further developed and used to automate the treatment of complex geometries for input into DTNS3D.

Accomplishment Description

The DTNS series of computer codes was developed at David Taylor Research Center for the numerical solution of the Reynolds-averaged Navier-Stokes equations for incompressible flow. The equations are discretized using an upwind differenced, total-variation-diminishing scheme for the convective terms, along with central differencing for the diffusion terms. Complex geometries, which are split into multiple blocks, are readily computed with the codes because the block boundary information consists only of single-line inputs specifying which blocks interface with each other. During the 1989-90 operational year the NAS facility was used in the validation of the DTNS3D code through the computation of the flow about several appended submarine geometries developed for the DARPA SUBOFF project. In one of the more complex cases, a 29-block calculation was performed for a submarine with a sail and four stern planes. A computation

typical of those carried out to date used about 12 megawords of memory and up to 10 hours of Cray-2 time for solutions using about 500,000 grid points.

Significance

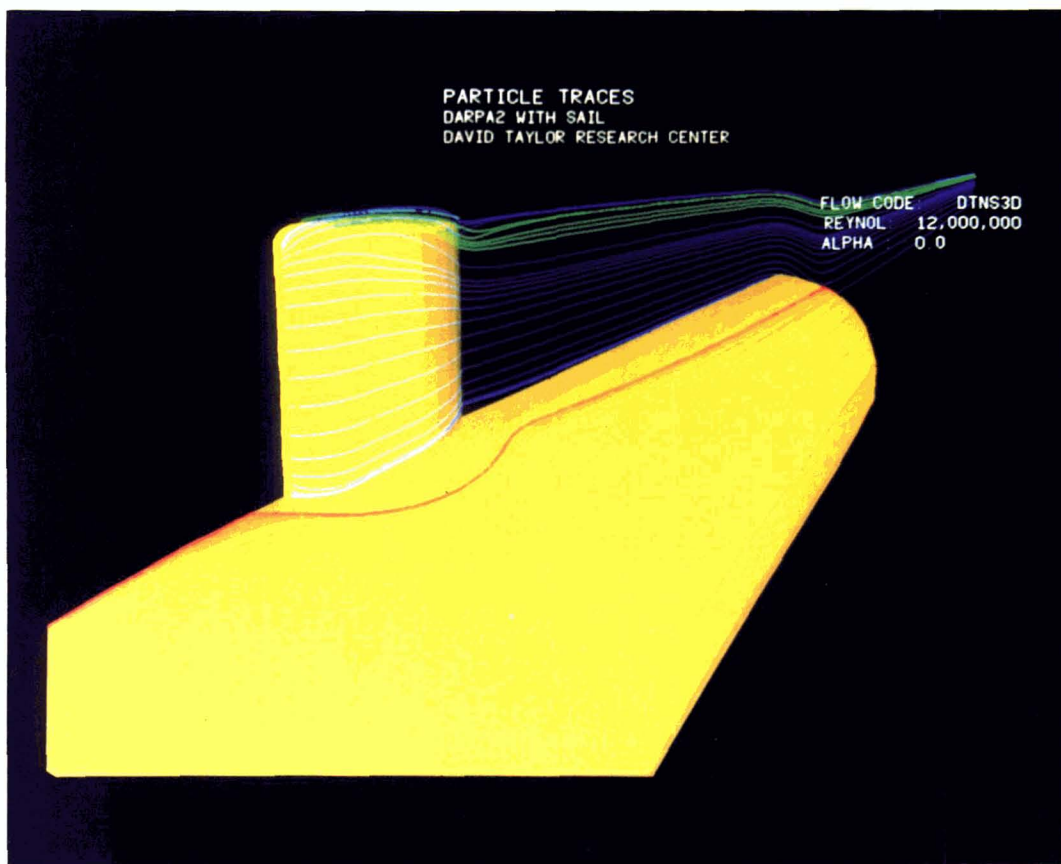
The development of the DTNS codes represents a significant step forward in the development of a robust capability to solve the flow equations for a wide variety of complicated naval geometries. The power of the multiblock approach has been demonstrated on the DARPA SUBOFF bodies. Initial progress toward an unsteady version brings within reach a very general unsteady computational capability. The emphasis on the usability and reliability of these codes means it is now easy to calculate the flow over a wide variety of geometries, and the codes are being used by researchers in fluid dynamics who are not interested in code development.

Future Plans

Research on boundary-fitted coordinate systems will continue to focus on the development of improved control mechanisms for coordinate representation. Improved turbulence models will be developed and implemented. Validation of improved flow codes will receive high priority, with computed and experimental results being compared. Development of methods for the solution of the unsteady equations will continue.

Publications

Haussling, H. J.; Gorski, J. J.; and Coleman, R. M. "Computation of Incompressible Fluid Flow about Bodies with Appendages." *Proceedings of the International Seminar on Supercomputing in Fluid Flow*. Lowell, MA, Oct. 1989.



Particle traces on a DARPA2 submarine with sail.

CERIAL FIVE
COLOR PHOTOGRAPH

An Analysis of the Viscous Flow through a Compact Radial Turbine by the Average-Passage Approach

James D. Heidmann, Principal Investigator
NASA Lewis Research Center

Research Objective

The objective of this study is to gain an understanding of the complex flow physics associated with the flow through a compact radial turbine. An additional objective is to extend the use of the average-passage code and validate its application to radial turbomachinery.

Approach

Three-dimensional Euler and Navier-Stokes codes based on the average-passage set of equations for turbomachinery are used with H-grids to calculate the rotor flow field. Blade-clearance and hub-rotation effects are modeled in the viscous calculation.

Accomplishment Description

The three-dimensional average-passage code was modified for use with radial turbomachinery and was used to analyze the flow field in an advanced compact-radial-turbine rotor. The computational solutions improved progressively; they included an Euler solution, a Navier-Stokes solution without clearance, and a Navier-Stokes solution with clearance. The Navier-Stokes solution with clearance predicts secondary vortical flows, which help explain experimental trends in the rotor exit-flow angle, and it also best predicts the experimental distribution of loss from hub to tip at the rotor exit. The viscous

calculation requires about 4 Cray-2 hours and 2.5 megawords of memory.

Significance

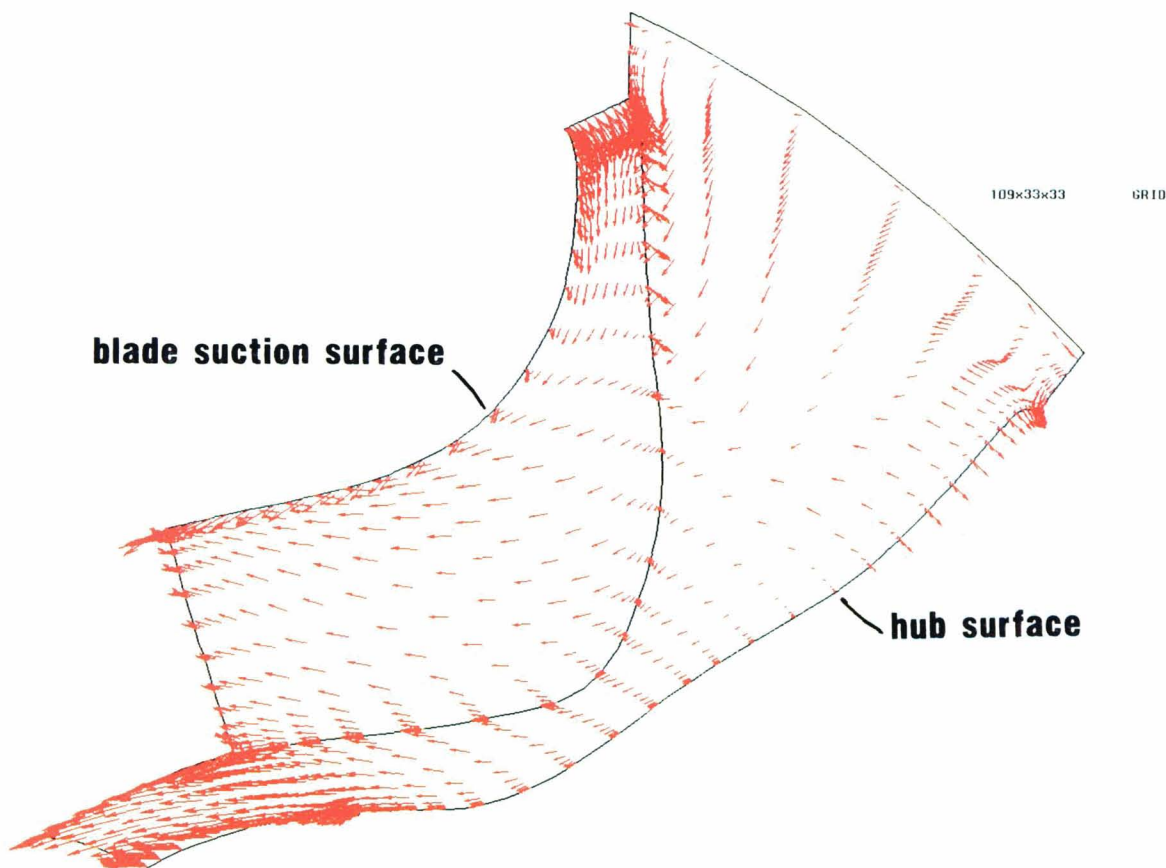
The average-passage code allows for the interpretation of secondary flows. It also brings a better insight into the flow physics of turbomachinery, which is particularly complex in radial machines such as the compact radial turbine. This insight can lead to an explanation of experimental trends, which makes the code a powerful new analysis tool. With experience, the knowledge of secondary flow patterns can lead to improved turbomachinery designs and improved engine efficiencies.

Future Plans

The average-passage code will be applied to the compact radial turbine stage to determine the effects of the upstream stator on the rotor flow field, and the results will be compared with the previous rotor calculations and experimental data.

Publications

Heidmann, James D., and Beach, Timothy A. "An Analysis of the Viscous Flow Through a Compact Radial Turbine by the Average Passage Approach." NASA TM-102471 and ASME Paper 90-GT-64, June 1990.



Relative velocity vectors near the surface of a compact-radial-turbine rotor, viscous clearance solution; grid size $109 \times 33 \times 33$.

Direct Simulation of Turbulent Reacting Flows

James C. Hill, Principal Investigator

Co-investigators: Andy D. Leonard and Dana G. Haugli
Iowa State University

Research Objective

Simple chemical reactions in three-dimensional turbulent flows are being studied with direct simulations of the Navier-Stokes equations and mass conservation equations. The long-range objectives of the research are twofold: (1) to gain insight into the physical processes of turbulent mixing and chemical reaction, and (2) to use this insight to improve modeling methods for reacting flows. The immediate objective is to see how a uniform shear flow affects the structure of the reaction zones compared with the case of decaying, homogeneous turbulence.

Approach

A pseudospectral code was used to calculate the velocity components and species concentrations for a second-order, two-species reaction of non-premixed species in incompressible, homogeneous turbulence. Either decaying, homogeneous turbulence was used or uniform mean shear with a deforming metric was used.

Accomplishment Description

For the non-premixed reaction, regions of highest reaction rate occur where the compressive strain rate is greatest in both decaying, homogeneous turbulence and uniformly sheared turbulence. The eigenvectors for the compressive strain rate are perpendicular to the reaction zone in both cases (as shown in figure (a) for the sheared case). Since the peak strain rates occur in the tails of the strain-rate probability density function (PDF), it appears that rare events make a significant contribution to the overall reaction rate. The PDF of the alignment of the concentration gradient vector with principal strain rates shows that for both flows the reaction-zone alignment is determined by the compressive strain rate, but in the shear-flow case the PDF for the most extensional strain rate is flat. In the shear case, regions of highest reaction rate also tend to lie in axially strained vortices (as shown with the horseshoe vortex in figures (b) and (c)), contrary to the secondary role of vorticity in the isotropic case. Each 64^3 run with eight scalars requires about 8 megawords of memory and 3 hours of CPU time on the Cray-2. Shear-flow runs of size $128 \times 128 \times 64$ with three scalars require 16 megawords of memory and 30 seconds per time step (8 to 10 hours per run).

Significance

This work has shown that rare fluid mechanics events make important contributions to reaction rates, and that vortices (or "braids" in mixing layers) are the sites of high reaction rates in shear flows. Consequently, theories involving turbulence intermittency may be important for modeling reactions in homogeneous turbulence, and mechanistic models involving axially strained vortices may suffice for reactions in shear turbulence.

Future Plans

The mixing study is being continued with the development of forcing algorithms to decouple the time scales for reaction from the time scales for the decay of the turbulence. More extensive shear-flow calculations are also under way. In addition, new code is being developed that will enable the simulations to accommodate higher strain rates and reaction rates.

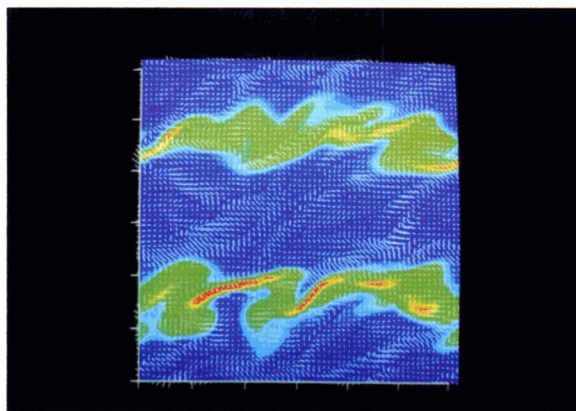
Publications

1. Leonard, Andy D., and Hill, James C. "Turbulent Structures and Local Reaction Rates in Homogeneous Turbulent Flow." Presented at Turbulence 89: International Conference on Organized Structures and Turbulence in Fluid Mechanics, Grenoble, France, Sept. 1989.
2. Leonard, Andy D., and Hill, James C. "The Influence of Local Kinematics on Chemical Reactions in Homogeneous

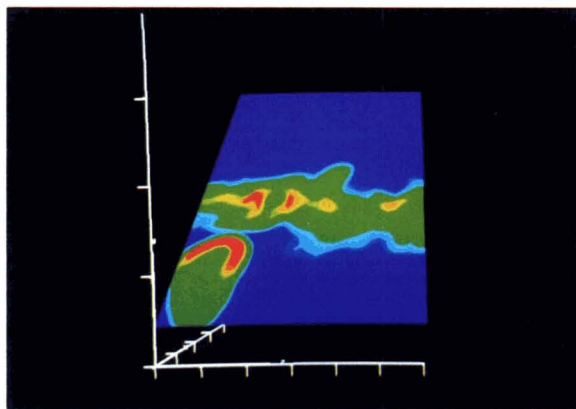
Turbulence." Paper #11h, American Institute of Chemical Engineers 1989 Annual Meeting, San Francisco, Nov. 1989.

3. Leonard, Andy D., and Hill, James C. "Numerical Studies of Scalar Dissipation and Mixing in Turbulent Reacting Flows." Presented at the IUTAM Symposium on Fluid Mechanics of Stirring and Mixing, La Jolla, CA, Aug. 1990.

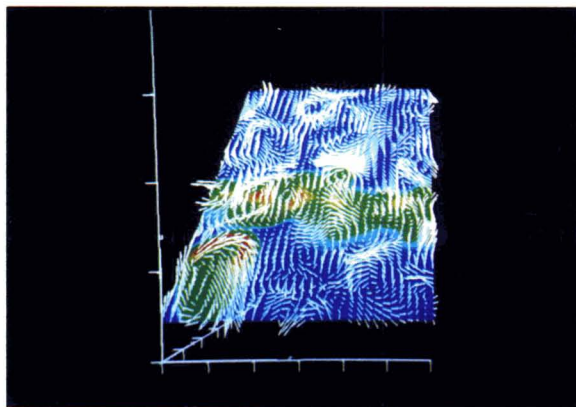
4. Leonard, Andy D., and Hill, James C. "Kinematics of the Reaction Zone in Homogeneous Turbulence." Presented at the Twelfth Symposium on Turbulence, Rolla, MO, Sept. 1990.



(a)



(b)



(c)

Reaction rate contours (highest rate in red) for homogeneous turbulent shear flow and non-premixed reactants. (a) Cross section showing both reaction zones and the eigenvectors (white lines) of the most compressive principal strain rate. (b) and (c) Cross section in inclined plane (26°) showing horseshoe-shaped structure, (b) without and (c) with vortex vectors superimposed.

A Computational and Experimental Parametric Study of Three-Dimensional Side-Wall Compression Scramjet Inlets at Mach 10

Scott D. Holland, Principal Investigator
Co-investigator: John N. Perkins
North Carolina State University

Research Objective

To demonstrate that computational fluid dynamics is useful as a design tool for supersonic combustion ramjet inlets in a hypersonic environment.

Approach

A preliminary computational parametric study is performed to eliminate from consideration inefficient designs; this leads to an "optimized" configuration. A wind tunnel model of the proposed configuration is then constructed to provide experimental verification of the design. The class of inlet chosen for the study was the three-dimensional side-wall compression inlet. Since there exists an experimental and computational data base (though sparse) in the supersonic regime, inlets of this class seemed the obvious choice for extension to the hypersonic parametric study. Additionally, the three-dimensional inlet affords a relatively simple, generic geometry, while producing a highly complex flow field dominated by shock/shock and shock/boundary-layer viscous interactions.

Accomplishment Description

A three-dimensional Reynolds-averaged Navier-Stokes code was modified for the current computational parametric study to identify the effects of leading-edge sweep and cowl position. Additionally, the effects of boundary-layer state (laminar/turbulent) and thermal wall boundary conditions (adiabatic/300-K constant wall) were assessed. The code solves the governing equations in full conservation form using MacCormack's time-asymptotic, explicit, predictor-corrector method. This method is second-order accurate in time and space and yields to a high degree of vectorization. The figure shows a lengthwise cross section of the inlet, revealing parti-

cle traces on the side wall and forebody plane (top of figure). Multiple regions of shock-induced separation and reattachment demonstrate the complexity of the flow. The computation required approximately 4 Cray Y-MP hours and 3 megawords of memory.

Significance

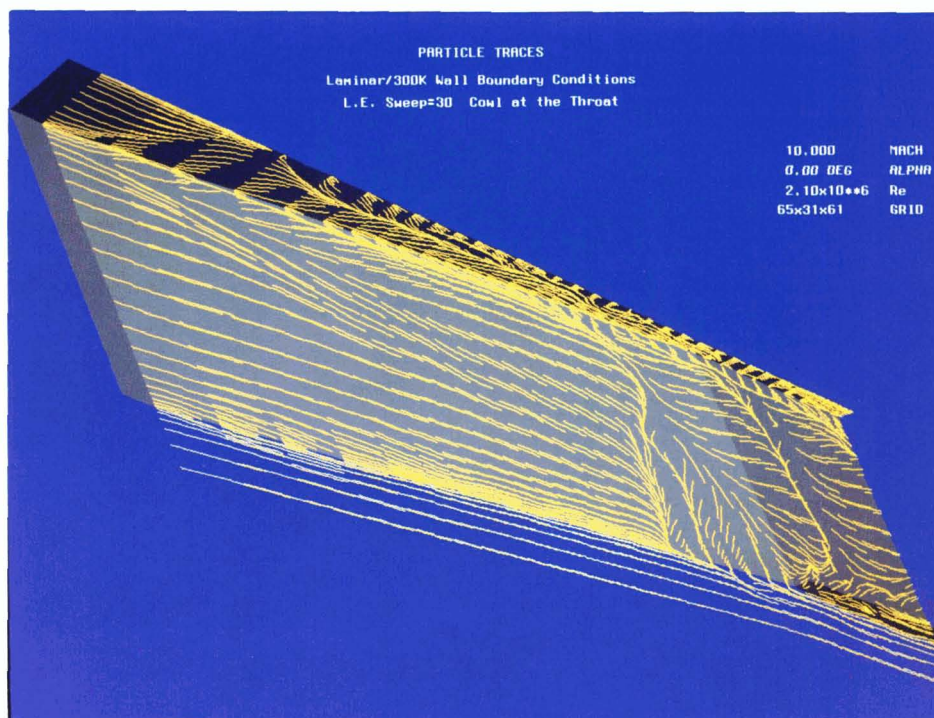
Proposed hypersonic vehicles will operate in flight regimes that impose high thermal as well as aerodynamic loads. To relieve some of these aerothermal loads, it is desirable to sweep the sharp leading edges of the inlet side wall. The degree to which this sweep affects the internal aerodynamics of the inlet was assessed. When comparisons were made of mass capture, throat Mach number, total pressure recovery, and average inlet compression ratio at different sweeps, it was found that the effect of sweeping the leading edge from 0° to 30° was negligible. Further increase in sweep tends to degrade the mass capture and compression while improving the total pressure recovery and throat Mach number. A trade-off of these parameters leads to a 45°-sweep configuration.

Future Plans

A project parallel to the experimental verification of these findings has evolved. While preparing for the experimental work, an opportunity arose to test a model of this class at Mach 6 in CF₄. Since CF₄ is neither calorically nor thermally perfect, the code will be extended to handle the virial gas equations, allowing benchmark comparisons with virial (real-gas) data.

Publications

Holland, Scott D., and Perkins, John N. "A Computational Parametric Study of Three-Dimensional Sidewall Compression Scramjet Inlets at Mach 10." AIAA Paper 90-2131, July 1990.



Particle traces on a three-dimensional side-wall compression inlet with leading-edge sweep = 30 and cowl at the throat, in laminar/300-K wall boundary conditions; $M = 10$, $\alpha = 0^\circ$, $Re = 2.10 \times 10^6$, grid size $65 \times 31 \times 61$.

Turbulence Modeling for Compressible/Hypersonic Flows

C. C. Horstman, Principal Investigator

Co-investigators: J. R. Viegas, T. J. Coakley, and P. Rodi
NASA Ames Research Center

Research Objective

To develop, test, and validate turbulence models for complex aerodynamic flow fields at supersonic and hypersonic speeds.

Approach

Three-dimensional Navier-Stokes codes with various levels of turbulence modeling are used to predict various flow fields that have been experimentally documented.

Accomplishment Description

Solutions were obtained for a series of three-dimensional, shock-wave/turbulent-boundary-layer interaction flow fields using several turbulence models. As the shock strength increased, the peak values of measured skin friction were underpredicted by as much as 52%. No turbulence model correctly predicted all the test cases. The simpler algebraic models outperformed the more complex two-equation eddy-viscosity models. The use of wall functions caused significant underpredictions of the interaction size. The accompanying figure shows the flow geometry, along with typical results, for a 16° sharp fin at Mach 4. A single computation needed about 20 Cray Y-MP hours and 10 megawords of memory.

Significance

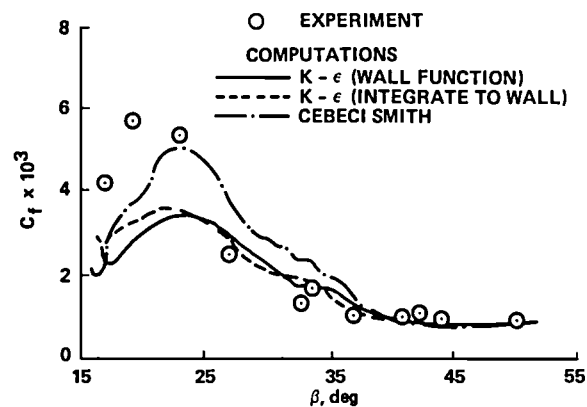
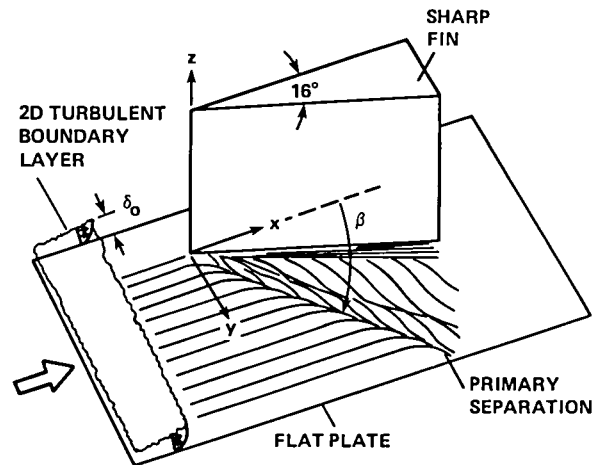
Proposed aerospace vehicles will sometimes operate in flight regimes that are beyond the accessibility of ground-based wind tunnels. Therefore, the vehicle design process will depend on computational fluid dynamics. For accurate flow simulation, valid turbulence models must be used.

Future Plans

Work will continue to improve the turbulence modeling, grid resolution, and boundary condition treatment. Extension to higher Mach numbers is also planned.

Publications

Kim, K.-S.; Lee, Y.; Alvi, F. S.; Settles, G. S.; and Horstman, C. C. "Laser Skin Friction Measurements and CFD Comparison of Weak-to-Strong Swept Shock/Boundary-Layer Interactions." AIAA Paper 90-0378, 1990.



Flow geometry and typical results for a 16° sharp fin; $M = 4$.

Prediction of Vortical Flows Using Incompressible Navier-Stokes Equations

C.-H. Hsu, Principal Investigator
ViGYAN, Inc./NASA Langley Research Center

Research Objective

To develop a computational method for calculating low-speed, three-dimensional viscous flow fields.

Approach

A computer code VOR3DI is developed. The code uses the implicit upwind-relaxation finite-difference algorithm with a nonsingular eigensystem to solve the preconditioned, three-dimensional, incompressible Navier-Stokes equations in curvilinear coordinates.

Accomplishment Description

Steady-state Navier-Stokes solutions were carried out for three low-aspect-ratio wings with rounded edges. Computed surface-particle traces at $\alpha = 20^\circ$ show the flow separation and attachment resulting from the generation and interaction of leading-edge vortices.

Significance

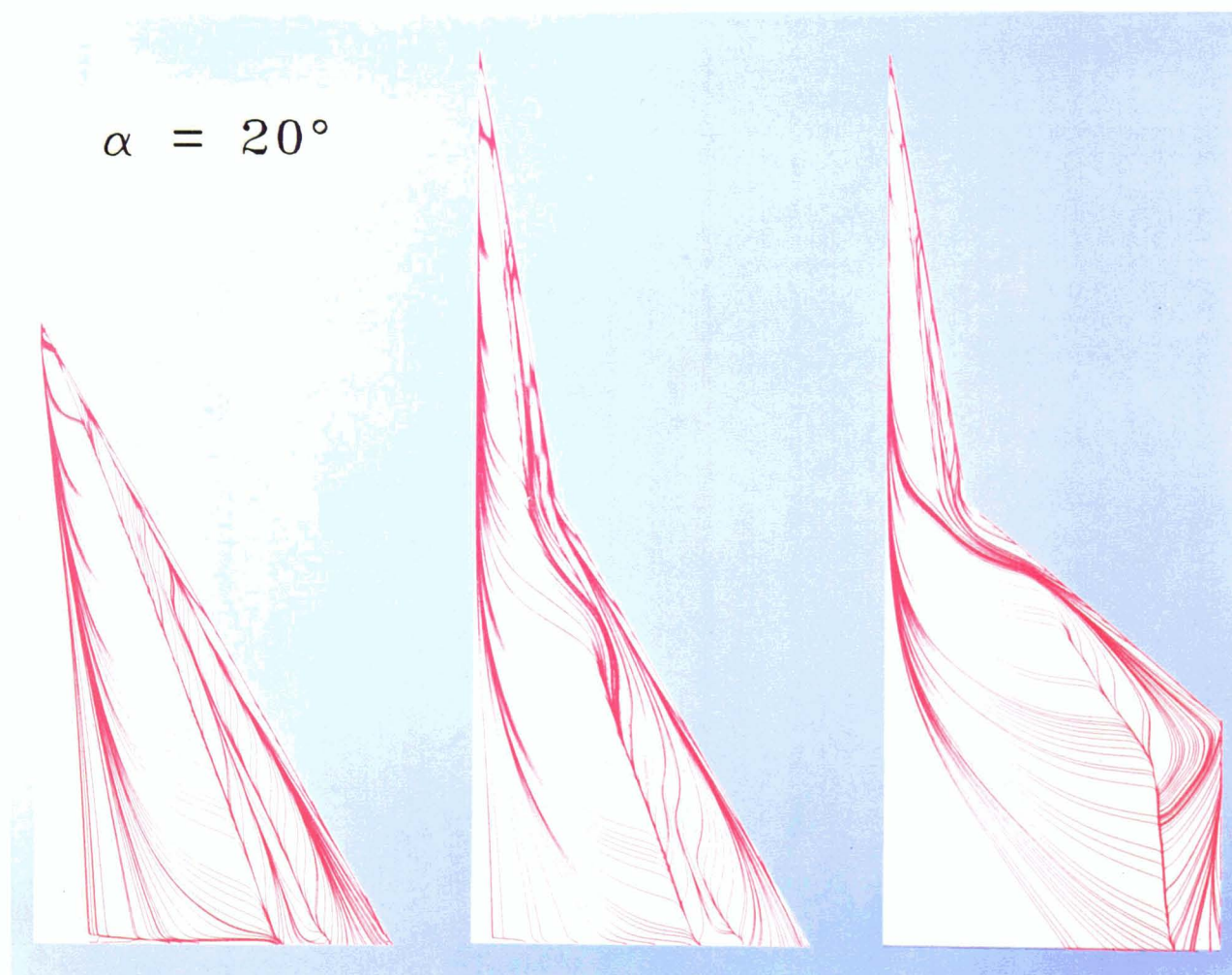
Computations for highly three-dimensional, leading-edge vortex flows over a delta wing, a double-delta wing, and a cropped double-delta wing have shown good agreement with experimental data. The analyses indicate that the present code can predict important features of complicated three-dimensional vortical flow fields.

Future Plans

The computer code has been extended, to be able to make time-accurate computations. Numerical simulations of vortex breakdown will be performed.

Publications

1. Hsu, C.-H., and Liu, C. H. "Numerical Study of Vortical Flow over a Sideslipping Delta Wing." AIAA Paper 90-3001-CP, Aug. 1990.
2. Hsu, C.-H.; Chen, Y.-M.; and Liu, C. H. "Preconditioned Upwind Methods to Solve 3-D Incompressible Navier-Stokes Equations for Viscous Flows." AIAA Paper 90-1496, June 1990.
3. Hsu, C.-H., and Liu, C. H. "Prediction of Vortical Flows on Wing Using Incompressible Navier-Stokes Equations." *Proceedings of the Third International Congress of Fluid Mechanics, Vol. 2*. Cairo, Egypt, Jan. 1990, pp. 573-586.
4. Hsu, C.-H., and Liu, C. H. "Simulation of Leading-Edge Vortex Flows." To be published in *Theoretical and Computational Fluid Dynamics 2* (1990).
5. Hsu, C.-H., and Liu, C. H. "Navier-Stokes Computation of Flow Around a Round-Edged Double-Delta Wing." Submitted to AIAA J. 28 (1990).



Particle traces on a delta wing, a double-delta wing, and a cropped double-delta wing; $\alpha = 20^\circ$.

Turbulence Modeling for Submarine Flow Field Computation

Thomas T. Huang, Principal Investigator

Co-investigator: Yu-Tai Lee

David Taylor Research Center

Research Objective

To examine the effect of the turbulence model on the solutions of the Navier-Stokes solvers and submarine flow fields.

Approach

Three-dimensional, steady-state, incompressible, Reynolds-averaged Navier-Stokes equations are solved using a finite-volume formulation based on a central-difference spatial discretization and an explicit Runge-Kutta time-stepping scheme. Algebraic numerical grid generation is used.

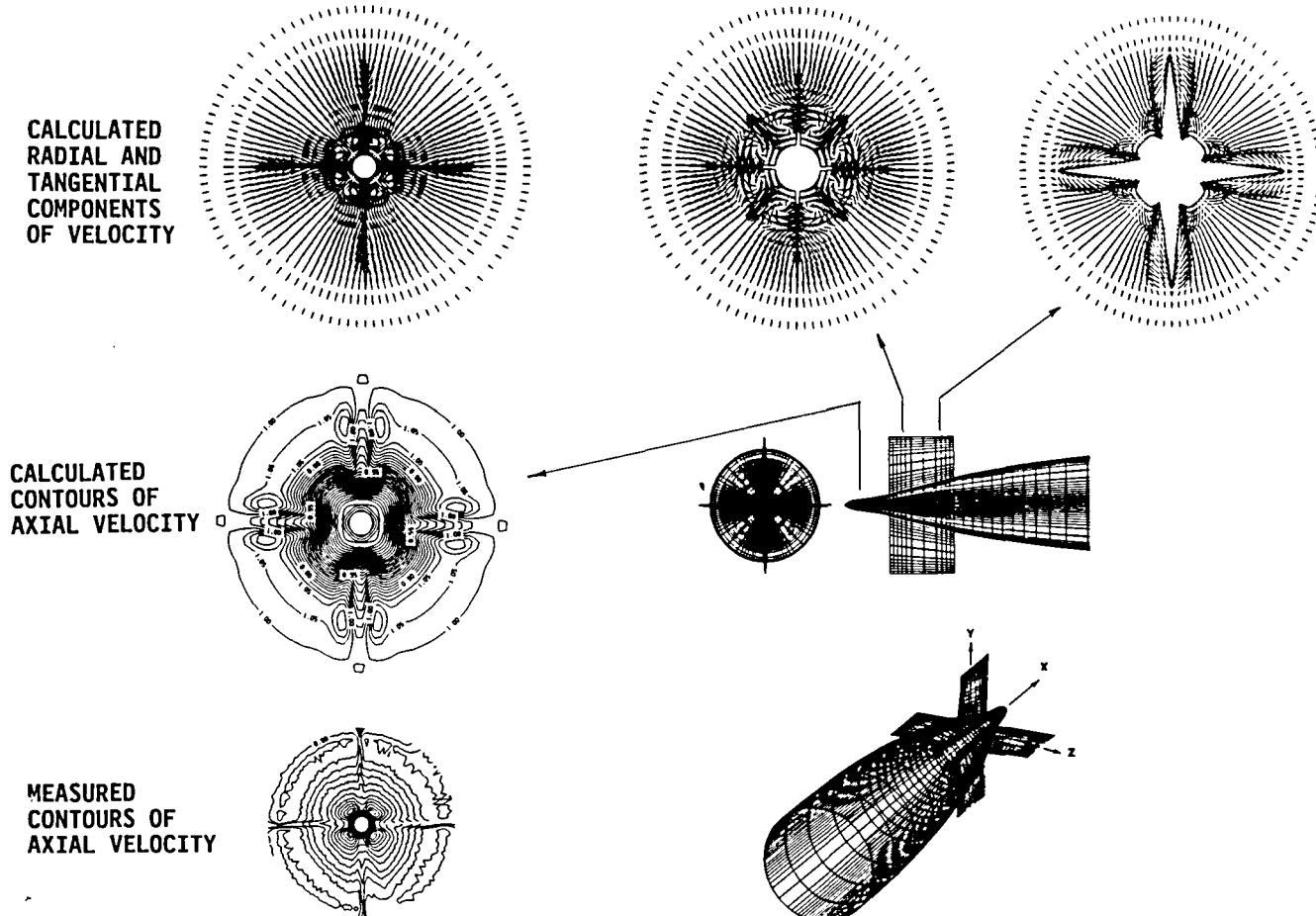
Accomplishment Description

Laminar/turbulent flows past an axisymmetric body and a body of revolution with appendages in the tail region were predicted at a 0° angle of attack. Calculated results for turbulent axisymmetric flows at a Reynolds number of 6.6×10^6 agreed well with the experimental data. The accompanying figure shows the stern region of the computational grid for an appended submarine. The predicted laminar flow on a generic appended body of revolution at a Reynolds

number of 0.5×10^6 , shown in the figure, was found qualitatively to be in good agreement with the turbulent-flow measurements on a body of revolution with slightly different appendages. The results for the appended body of revolution were obtained using a coarse ($43 \times 13 \times 13$) grid for the initial solutions and a fine ($85 \times 25 \times 25$) grid for the final solutions. The running times of each iteration for the coarse and the fine grids are 0.097 and 0.639 Cray Y-MP seconds, respectively. Three thousand iterations are made on the coarse grids and 2000 on the fine grids. The total CPU time for the case is 27 minutes. This work was published in the proceedings of the Sixth International Conference on Numerical Methods in Laminar & Turbulent Flow, Swansea, U.K., July 1989.

Significance

The capability developed from this project will lead to a significant increase in the understanding of submarine hydrodynamics and to an improvement in ship design.



Flow field at a submarine stern.

Numerical Simulation of an F-16A at Angle of Attack

Gary W. Huband, Principal Investigator
Co-investigators: J. S. Shang and Michael J. Aftosmis
Wright Research and Development Center

Research Objective

The objective of this project is to show that realistic, complex aircraft can be accurately simulated in flight using computational fluid dynamics methods. Specific goals are to highlight the current simulation capability and to identify areas that need improvement.

Approach

An F-16A fighter at an angle of attack and transonic Mach number was simulated by solving the mass-averaged Navier-Stokes equations on a single-block grid. MacCormack's explicit predictor-corrector algorithm was used to integrate the equations of motion, and a modified version of the Baldwin-Lomax algebraic turbulence model was used to achieve turbulent closure.

Accomplishment Description

A numerical simulation of a wind tunnel experiment was performed for an F-16A at a free-stream Mach number of 0.85, a Reynolds number of 12.75×10^6 , and an angle of attack of 16° . The computed lift coefficient differed from the wind tunnel lift coefficient by less than 1%, and the computed drag coefficient differed from the wind tunnel drag coefficient by 8%. The computed surface pressure coefficients along the body, wing, and empennage also matched wind tunnel data reasonably well. New grid generation methods were developed and used that ensure a smooth flow-field grid, but still accurately describe the body shape. These grid generation methods

also significantly reduce the time to generate a grid about a complex three-dimensional shape.

Significance

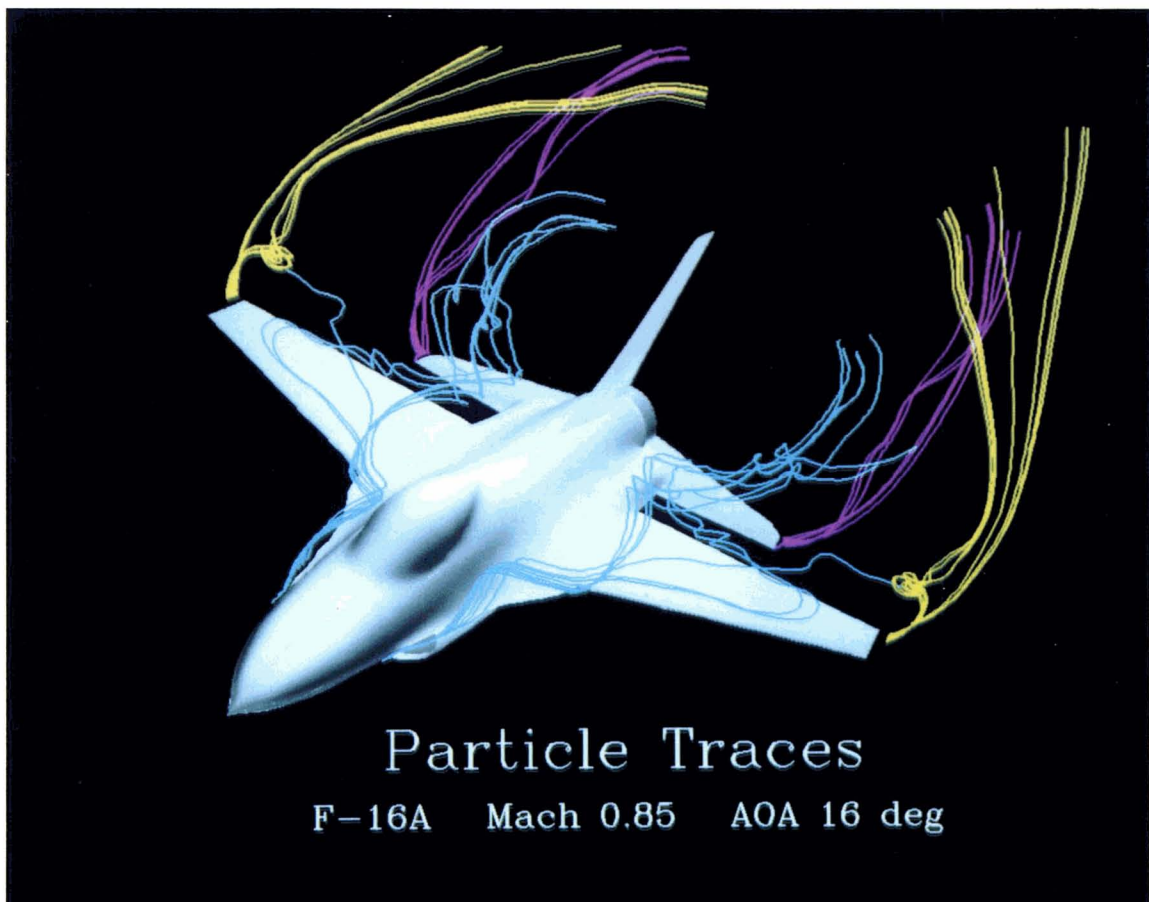
This is a first step toward the development of an aircraft design tool that can provide detailed and accurate information about a configuration's aerodynamics. Such a tool will reduce design time, wind tunnel time, and flight test time, while allowing the development of more innovative designs with lower risk.

Future Plans

Three areas are targeted for future research—increasing code efficiency, improving the turbulence model in separated regions, and improving the grid generation methods. Because the time to obtain a solution must be reduced as much as possible, an implicit code is under consideration. To ease the grid generation requirements, multiblock solvers and unstructured solvers are also being considered.

Publications

1. Huband, G. W.; Rizzetta, D. P.; and Shang, J. S. "Numerical Simulation of the Navier-Stokes Equations for an F-16A Configuration." *AIAA Aircraft J.* 26, no. 7 (Jul. 1989): 634-640.
2. Huband, G. W.; Shang, J. S.; and Aftosmis, M. J. "Numerical Simulation of F-16A at Angle of Attack." Presented at the AIAA 28th Aerospace Science Meeting, Reno, NV.



Particle traces on an F-16A; $M = 0.85$, $\alpha = 16^\circ$.

Computation of Steady Three-Dimensional Separation

Ching-mao Hung, Principal Investigator
NASA Ames Research Center

Research Objective

To study several basic issues of steady three-dimensional flow separation, such as open versus closed separation, the existence of secondary separation, and the accessibility of three-dimensional separation.

Approach

The compressible Navier-Stokes equations are solved for supersonic flows over a sharp and a flat-faced blunt fin mounted on a flat plate. A finite-volume algorithm of MacCormack's explicit-implicit scheme is used. Separation of the boundary layer on the flat plate is investigated for various grid refinements and fin wedge angles. Results are compared with experimental measurements of surface pressure, to determine the accuracy of the computations.

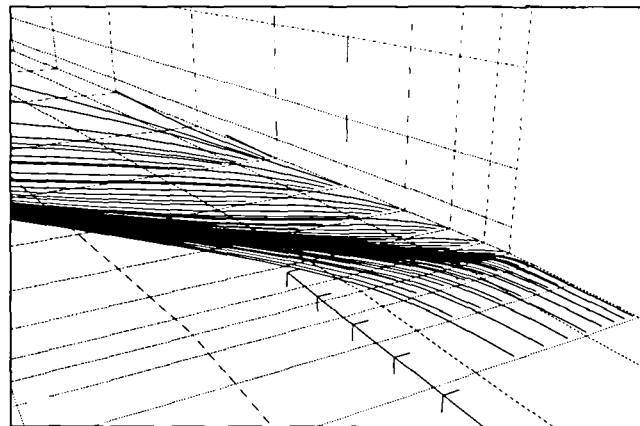
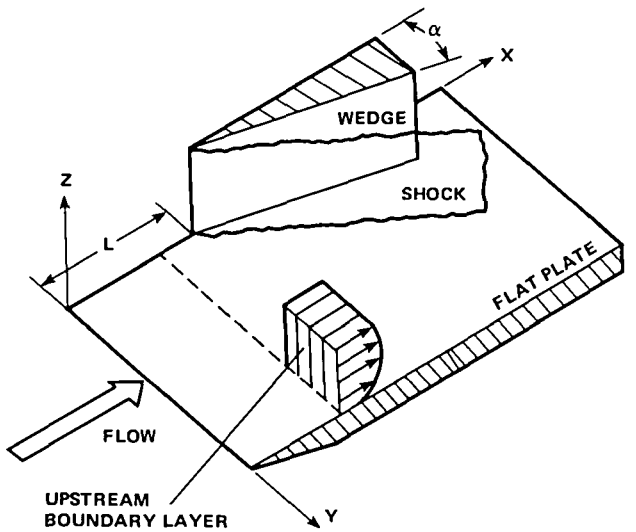
Accomplishment Description

(1) The results of using the same number of grid points with three different grid spacings show that the origin of the line of separation changes from an open-type to a closed-type separation. This demonstrates that the grid resolution can affect the "calculated" topology. (2) Surface particle traces show that there is no secondary separation. Another oil-accumulation line is just a demarcation between regions of high and low

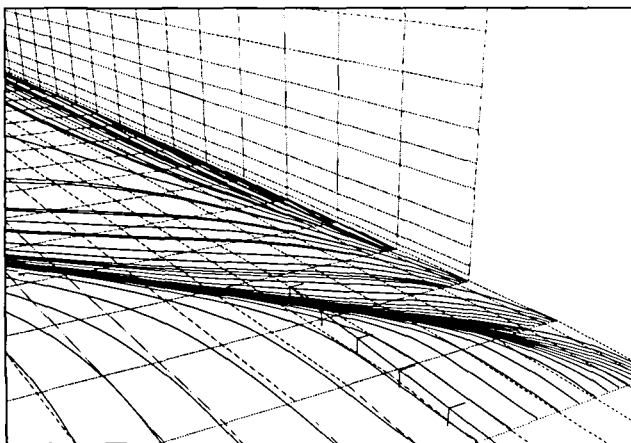
surface skin friction. (3) A close examination of the particle traces near the apex shows that the flow is already separated on the plate before reaching the conventional criterion of incipient separation, which is that the turning angle of the skin-friction line is equal to the shock angle. (4) The results demonstrate that flow particles above the body are able to access the separation region through the attachment node and the spiral nature of the separation surface.

Significance

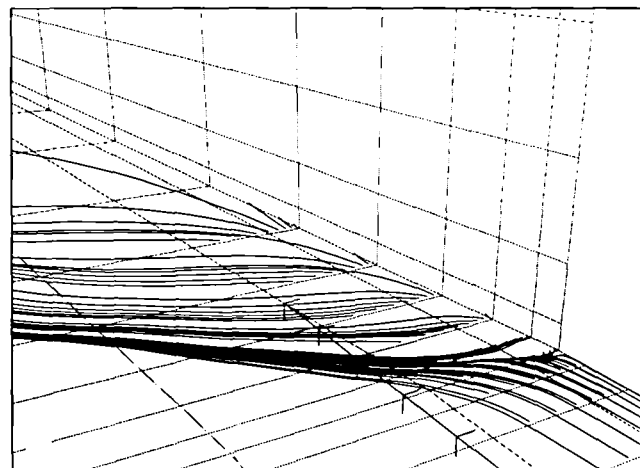
Some observations and new interpretations from this study that will have significant impacts on the understanding of the three-dimensional flow field are as follows: (1) Some of the numerically observed open-type separation is a result of insufficient grid resolution, and similarly, some of the experimentally observed open-type separation is the result of the low resolution of the device and facility. (2) An accumulation of oil flow on the surface is not necessarily a line of separation, as is usually inferred. (3) The conventional criterion of boundary separation, which involves comparing the turning angle of the skin-friction line to the shock angle, is not uniformly valid. (4) There is no steady three-dimensional separation that is totally closed by a separation surface as a bubble. All three-dimensional separation is accessible.



MEDIUM



COARSE



FINE

Grid refinement near the fin apex shows the simulated flow topology changes from an open-type to a closed-type separation.

Eddy-Resolving Model of the Pacific Ocean

Harley E. Hurlburt, Principal Investigator

Co-investigators: Alan J. Wallcraft and Jimmy L. Mitchell

Naval Oceanographic and Atmospheric Research Laboratory/JAYCOR

Research Objective

To develop, understand, and validate eddy-resolving models of the Pacific Ocean. This is a step in developing an eddy-resolving, global ocean-monitoring and -prediction system for the U.S. Navy. This system will rely heavily on supercomputers (Class VII and greater) and on satellite data, especially altimetry from satellites like GEOSAT, TOPEX/POSEIDON, and ERS-1.

Approach

We used primitive-equation layered models with two to four layers and $1/2^\circ$ to $1/8^\circ$ resolution. The versions used were hydrodynamic, thermodynamic, reduced-gravity (lowest layer infinitely deep and at rest), finite-depth flat bottom, and finite-depth with topography. A free surface was retained by treating gravity waves implicitly in the finite-depth versions and solving the resulting Helmholtz equations using the direct capacitance matrix technique. The models were spun up at coarser resolution, then continued with increased resolution forced by the Hellerman-Rosenstein monthly wind climatology. The different versions of the model were used to help sort out the dynamics and mechanisms behind different oceanic phenomena.

Accomplishment Description

We developed $1/2^\circ$ and $1/4^\circ$ finite-depth and $1/8^\circ$ reduced-gravity models on the Cray Y-MP. The models successfully simulated many features of the Pacific Ocean, as illustrated in the figure. The figure shows the sea surface height (SSH) from the $1/4^\circ$ model with realistic topography. Surface flow tends to parallel isolines of SSH. Some of the salient features are the Subarctic Front (40–45N across the Pacific) and the Kuroshio Current/Kuroshio Extension (along the western boundary north

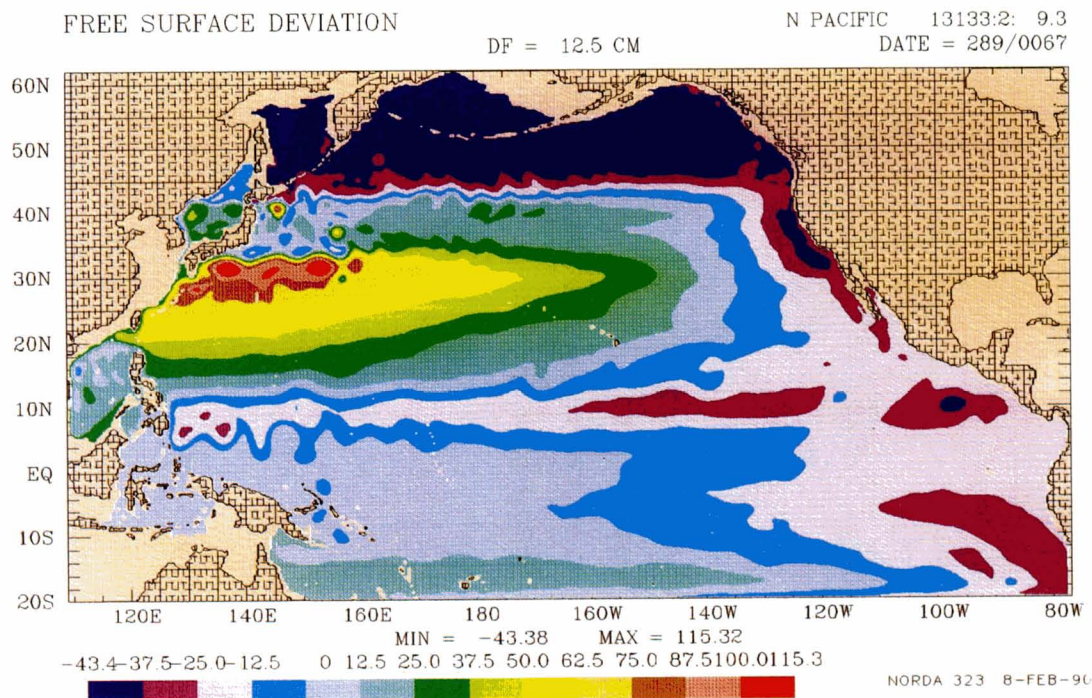
of 14N, then eastward across the Pacific at about 35N). The Kuroshio bifurcates at the Shatsky rise (about 160E) and a contour connects to the Subarctic Front, an observed feature that does not occur in the flat-bottom and reduced-gravity models. The ribbon of blue marks the North Equatorial Current flowing westward at 10–15N, the Mindanao Current flowing southward along the Philippines coast, and the eastward North Equatorial Countercurrent at about 5N. The Mindanao eddy is simulated southeast of the Philippines. Four-layer versions of the models also depict the westward equatorial surface current and eastward equatorial undercurrent, which are absent in the two-layer versions. We performed 24 Pacific simulations, integrating a total of 524 model years and using about 550 Cray Y-MP hours.

Significance

The Cray Y-MP at NASA Ames permitted the highest horizontal resolution ever in modeling the Pacific Ocean— $1/4^\circ$ for the finite-depth models and $1/8^\circ$ for a reduced-gravity model. This was a major step in developing an eddy-resolving, global ocean-prediction capability for the U.S. Navy, and it provided a high-impact demonstration of feasibility to the Navy. Applications include antisubmarine warfare, long-range weather prediction, climate monitoring and prediction, optimum-track ship routing, search and rescue, fisheries, and ice prediction.

Future Plans

We will continue to develop eddy-resolving, basin-scale and global-scale ocean models for ocean prediction using the Navy's Cray Y-MP 8/128 scheduled for installation at the Naval Oceanographic Office by October 1990.



Sea-surface height anomaly (cm) on a day in mid-October, year 67 of the model integration. The model is two-layer hydrodynamic with $0.25^\circ \times 0.35^\circ$ resolution (latitude, longitude) and realistic topography. The simulation depicts major current systems of the Pacific Ocean as well as current meanders and eddies caused by mesoscale flow instabilities. This simulation required 4.5 megawords of memory, 33 megawords of SSD, 2.5 Cray Y-MP hours per model year, and 72 Cray Y-MP hours for a 72-year integration, including spin-up at $1/2^\circ$.

Computational Advanced Propulsion Technology Research

Danny Hwang, Principal Investigator

Co-investigators: John Wolter, Joe Nenni, Mark Krein, and Dave Georgevich

NASA Lewis Research Center

Research Objective

To develop the ability to perform three-dimensional aerodynamic computations for high-speed nozzles and inlets. The verified computer codes will be used as design tools for analyzing new concepts even before a model is built.

Approach

Two three-dimensional Navier-Stokes codes, PARC3D and DLUM3D, were used for calculations. A three-dimensional grid generator by Soni, McClure, and Heikkinen was applied to create grids for analysis.

Accomplishment Description

The three-dimensional viscous calculations were performed for the AR410 transition duct by using PARC3D and DLUM3D. To reduce the uneven distribution of the boundary layer, caused by the skewness of H-grids on the surface, the decision was made to use an O-grid. The computer codes were modified to handle the O-grid. The preliminary laminar results of the secondary flow near the exit of the transition duct are shown in the

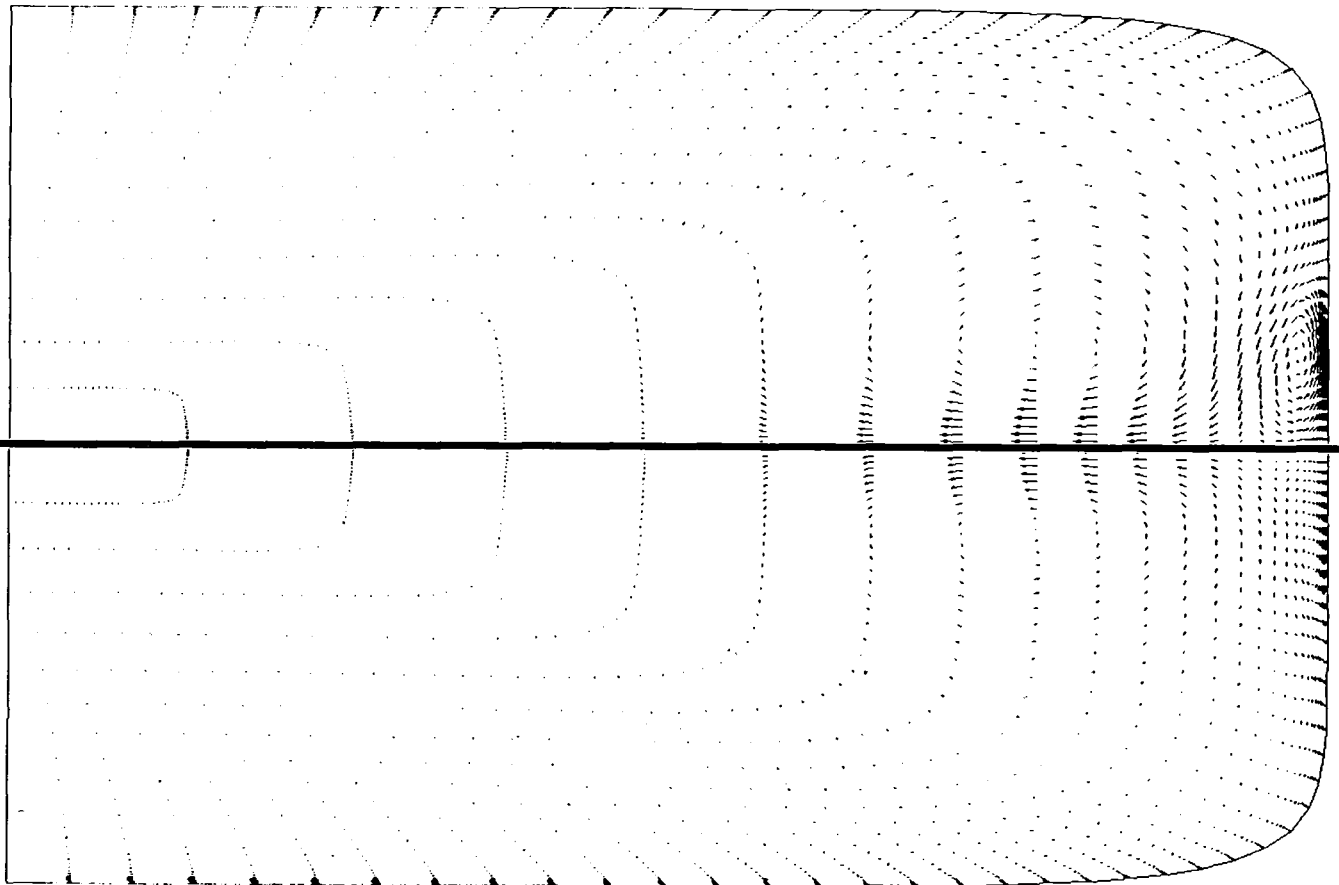
accompanying figure. The centers of the vortices obtained from the two codes agree fairly well. However, the vortices obtained from DLUM3D are stronger than those from PARC3D. A study is under way of turbulent flow, using the Baldwin-Lomax model and the $k-\epsilon$ model. The results from both laminar and turbulent flows will be compared with experimental data. Most of the computations required about 20 Cray-2 hours and 15 megawords of memory.

Significance

The total pressure loss, which can affect the performance of an advanced aircraft, can be more accurately predicted using viscous calculations. However, because of the long CPU time needed to carry out the viscous calculation, the Euler solution might be considered in some cases as a substitute.

Future Plans

The codes will be further used to study the effect of turbulence. By comparing the results with experimental data, we can determine the range of validity of each code.



Preliminary laminar results of the secondary flow near the exit of the AR410 transition duct; $M = 0.5$. (Top) DLUM3D solution. (Bottom) PARC3D solution.

Numerical Inversion of a Limited-Data X-Ray Transform

Steven H. Izen, Principal Investigator
Case Western Reserve University

Research Objective

The ultimate purpose of this research is to recover the three-dimensional density profile of supersonic gas flow between compressor blades in a wind tunnel. The more immediate goal of this work is to illustrate how well three-dimensional information about density profiles can be recovered when diffuse-illumination heterodyne holographic interferometry is used as an experimental technique, along with fully three-dimensional tomography for data analysis.

Approach

The data provided by diffuse-illumination heterodyne holographic interferometry experiments are line integrals of the index of refraction along lines perpendicular to the viewing direction of the holographic plate. Since the index of refraction is proportional to density, the data can be interpreted as determining the line integrals of density, or from a mathematical point of view, the data determine the x-ray transform of the density. Experimental constraints severely limit the viewing directions, so a mathematical inversion algorithm has been developed that takes advantage of the parallel-beam geometry inherent in the data acquisition. The algorithm transforms the experimentally obtained data to data at sample points in three-dimensional Fourier space in accordance with the projection-slice theorem. Then a singular value decomposition is performed to find the coefficients in a polynomial expansion for the density. Recent tests on the imaging of a laboratory flask have shown that this algorithm is able to recover significant three-dimensional information. Also, the tests have

indicated that extreme care is required in the lab to ensure that the data from each of the views of the hologram are properly registered. Each reconstruction requires 2 Cray-2 hours and 35 megawords of memory.

Significance

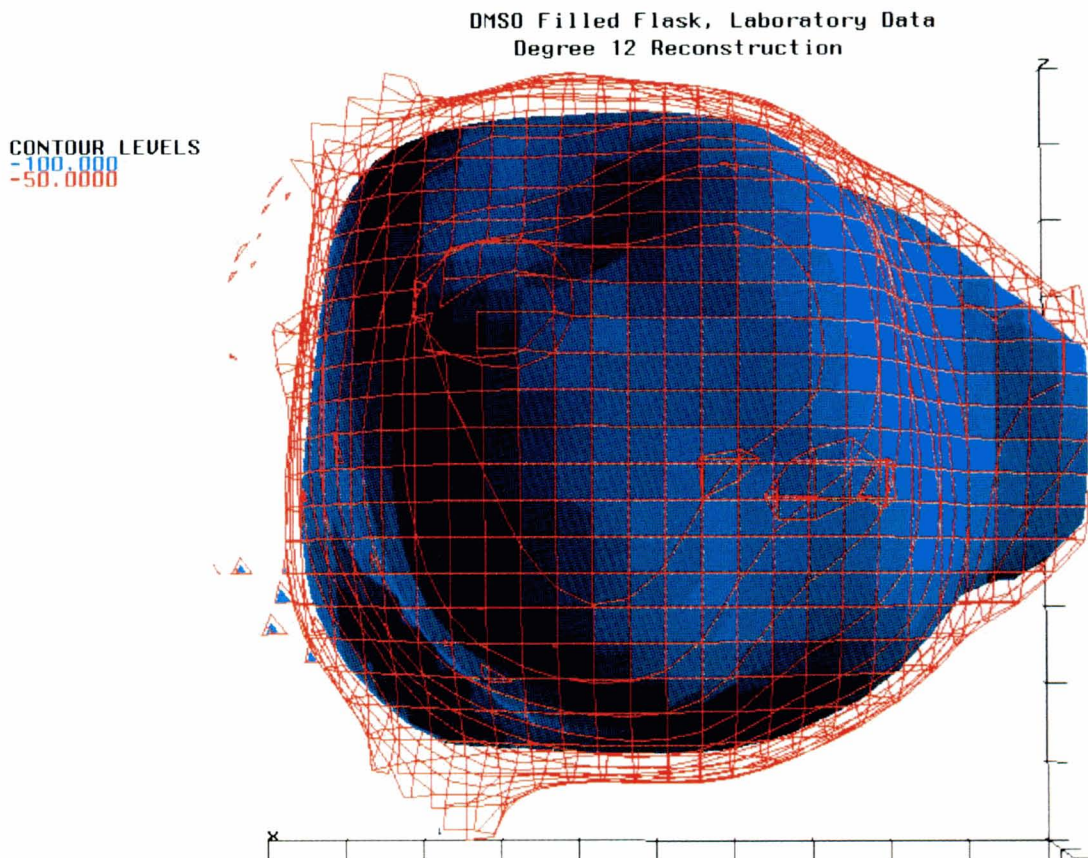
It is of interest to understand the behavior of flows in wind tunnel cascades. Computational fluid dynamics researchers have developed computer codes to model such flows. In order to verify these codes, experimental data are required. Current experimental techniques are able to supply only two-dimensional data, which are of limited use in the verification of three-dimensional models. The three-dimensional density profiles that will be computed from the diffuse-illumination heterodyne holographic interferometry data will provide verification data for the three-dimensional codes.

Future Plans

Enhancements such as edge-detection capability are being explored. Also, constraints arising from data generated by alternate experimental techniques are being incorporated to improve the reconstruction quality.

Publications

1. Izen, Steven H., and Decker, A. J. "Fully Three-Dimensional Tomographic Reconstructions from Holographic Interferometry Data." In preparation.
2. "An Application for a Limited Solid Angle X-ray Transform." *Proceedings of the 1989 AMS Conference on Radon Transforms and Tomography*.



Laboratory data for a DMSO filled flask; degree 12 reconstruction.

Linking Attractor Geometry to Turbulence Physics

Laurence R. Keefe, Principal Investigator
NASA Ames Research Center

Research Objective

To increase understanding of dynamical processes in turbulent shear flows by studying the structural characteristics of strange attractors, which have been demonstrated to underlie such flows.

Approach

Three-dimensional direct numerical simulation of low-Reynolds-number channel turbulence is used to calculate Lyapunov vectors and exponent spectra, fractal dimension, and other quantities suggested by chaos theory to be useful in analyzing dynamical events in such flows.

Accomplishment Description

Calculations were completed of Lyapunov exponent spectra from three different spatial resolutions of the channel flow at a single Reynolds number. The results show that these spectra, when scaled by the metric entropy and the number of non-negative exponents, have the same shape. The lowest resolution simulation yields an attractor of dimension ~ 360 , but using these scaling properties, we estimate that the true dimension of the attractor underlying flows in this computational domain is ~ 780 . The lack of increase in attractor dimension between the middle- and high-resolution cases indicates that the additional scales involved in the high-resolution case do not fundamentally affect the dynamics. Thus the effect of these scales should be assignable to an eddy viscosity or sub-grid-scale model. The scaling properties of the Lyapunov spectra can be seen in the figure. Each exponent calculation required 250 hours of CPU time and 16 megawords of memory.

Significance

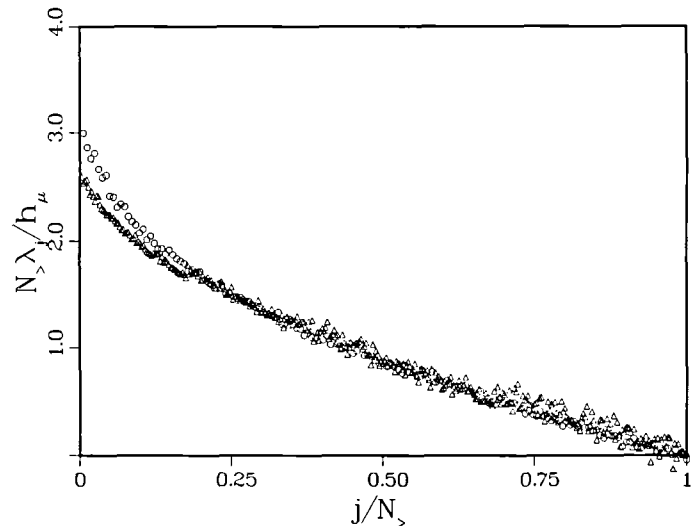
The strange attractor is an exciting model for the mathematical structure of fully developed shear flows. Exploration of the known structural characteristics of such attractors and their link with the phenomenology of fully developed flows will provide new insight into the dynamics of turbulence and new ideas for prediction and control.

Future Plans

We will examine the phase-space characteristics of "coherent structures" found by experimenters. What is the relation between attractor structure and turbulence modeling?

Publications

Keefe, Laurence; Moin, Parviz; and Kim, John. "Application of Chaos Theory to Shear Turbulence." In *The Ubiquity of Chaos*, ed. Saul Krasner. Washington, DC: AAAS, 1989.



Scaling properties of the Lyapunov spectra.

Solar Dynamic Space Power-System Design and Performance Analysis

John L. Klann, Principal Investigator
Analex Corporation/NASA Lewis Research Center

Research Objective

To accurately predict the performance of dynamic electric power modules in their intended space application and to evaluate the effects of design changes on such performance. Current emphasis is on the solar-powered Brayton cycle modules intended for Space Station Freedom.

Approach

The Closed Cycle Engine Program (CCEP) is being developed for the analysis of spacecraft electric power systems that use the Brayton thermodynamic cycle. The CCEP uses the same basic techniques as the Navy/NASA Engine Program (NNEP), a widely used computer program for the design and performance analysis of aircraft jet engines. However, CCEP is evolving with greater depth in its component design and performance simulations. In particular, it currently includes some transient orbital performance capabilities.

Accomplishment Description

Two large matrices are needed for radiosity calculations within the cavity of the power module's solar heat receiver, to properly evaluate the receiver's transient operating temperatures. Most of the incident solar flux within the heat receiver is absorbed by salt canisters that surround 82 equally spaced, gas heat-transfer tubes around the cylindrical internal wall. Active gas-flow length is 8 ft, and there are 96 canisters per tube. During the sunlit portion of an Earth orbit, part of the heat

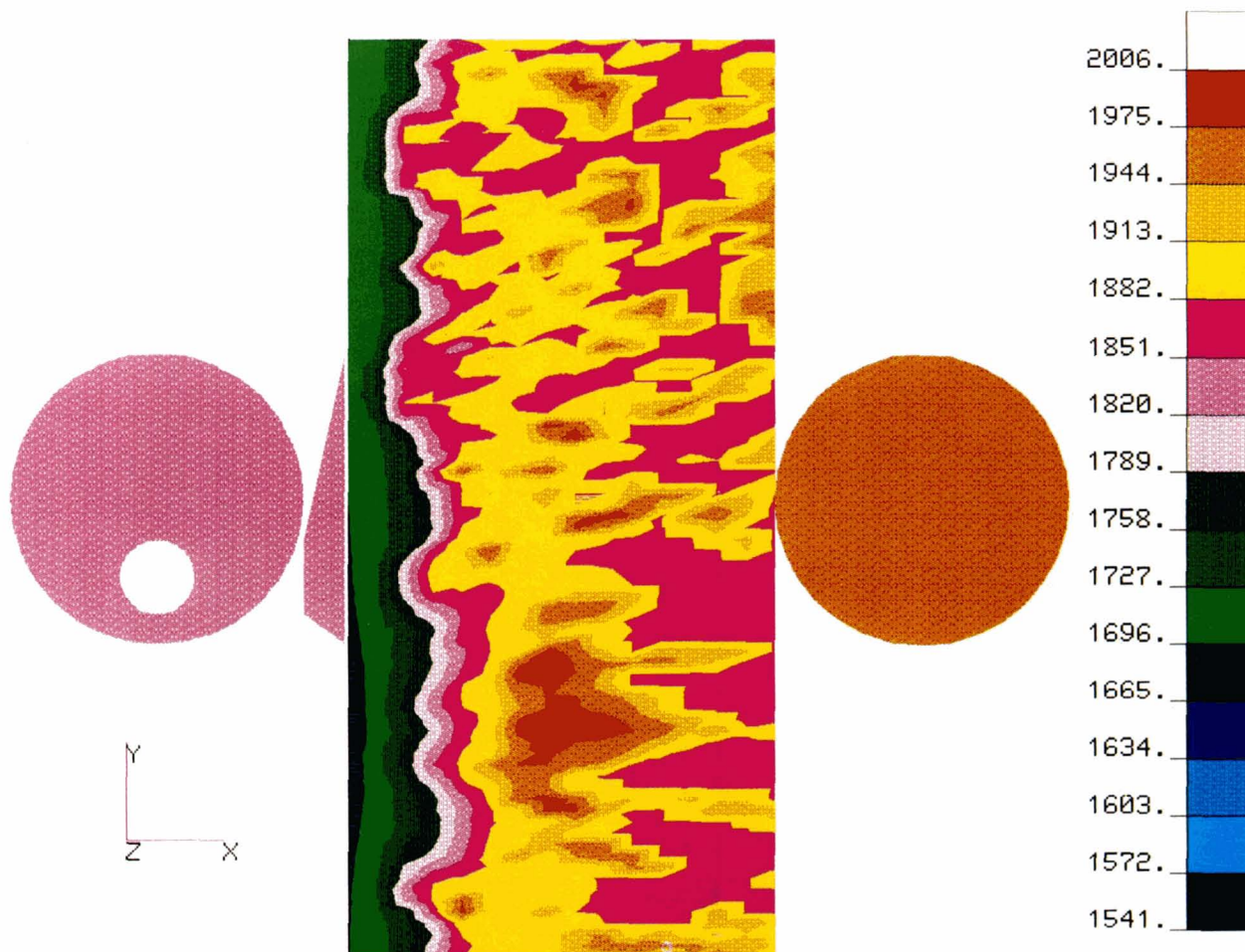
is stored by melting the salt for later use during passage through the Earth's shadow. Thus, heat is continuously provided to the working gas. The figure shows the highest predicted surface-temperature distributions that occur with the best flux-tailored mirror/receiver arrangement. Interior surfaces are unwrapped; the offset aperture is shown on the left, and the enclosing end on the right. The peak wall temperature was 1546° F (the figure is in °R) and occurred at sunset under minimum-insolation and orbital energy-equilibrium conditions (third orbit). Each orbital calculation requires 1 Cray-2 hour and about 4 megawords of storage.

Significance

The CCEP is an in-house development at NASA Lewis, and its capabilities are unique in the Space Station Freedom program. The NAS capabilities are needed to examine mirror/receiver designs so that receiver spatial and time-dependent temperature variations can be minimized. The results of this effort should maximize the life of this critical component.

Future Plans

The CCEP will continue to be exercised for component and system evaluations throughout the Phase-C effort for Space Station Freedom. The heat receiver radiosity calculations will be altered to accommodate assumptions other than a gray-body assumption, to evaluate the benefits of outer salt-canister surface treatments.



Receiver-wall temperature profile; N = 3, increment = 54.64.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Three-Dimensional Shock/Shock Interactions on the Inlet Side Wall

Ajay Kumar, Principal Investigator
Co-investigator: Dal J. Singh
NASA Langley Research Center

Research Objective

To numerically simulate the three-dimensional shock/shock interaction on the swept inlet side walls of the scramjet inlet at high Mach number.

Approach

The interaction of the inlet shock with the forebody shock is studied by solving three-dimensional, thin-layer Navier-Stokes equations. The governing equations are solved using van Leer's finite-volume flux-splitting algorithm. The impinging shock is treated as a sharp discontinuity across which exact shock jump conditions are applied. The inlet side wall of the scramjet inlet is modeled by a cylindrically blunted wedge, and the forebody shock by the extraneous impinging shock.

Accomplishment Description

The effects of shock impingement on the scramjet-engine-inlet leading edge are investigated numerically. Type IV and Type V interactions are investigated. Results of the present numerical investigation are compared with the available experimental data. For flow at Mach 5.94 and Reynolds number 202,500, the peak pressure is found to be 2.2 times and the peak heating 3 times the unimpinged stagnation values. The heating

rates and surface pressure are slightly lower for Type V interaction. The flow for Type IV interaction is found to be unsteady.

Significance

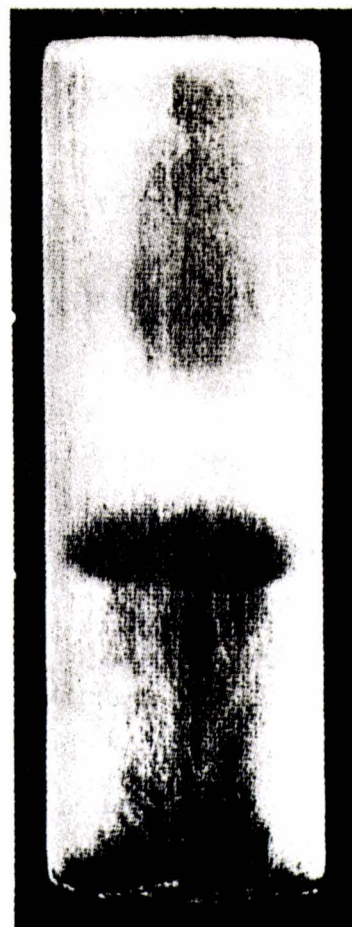
When the forebody shock interacts with the blunt inlet-leading-edge shock, a complex flow field is produced, with localized zones of high pressure and intense heating. The large temperature gradients cause thermal stresses that could result in structural failure. To determine the cooling requirements, pressure and heating rates on the body need to be predicted accurately.

Future Plans

Efforts are currently under way to investigate the unsteady behavior showed by the Type IV interaction. Also we plan to use multizone and/or adaptive grids to properly resolve the flow field.

Publications

Singh, D. J.; Kumar, A.; and Tiwari, S. N. "Three-Dimensional Shock-Shock Interactions of the Scramjet Inlet." AIAA Paper 90-0529, Jan. 1990.



Type V shock/shock interaction on the inlet side wall. (Left) Numerical simulation. (Right) Experiment.

C-2

ORIGINAL PAGE
COLOR PHOTOGRAPH

Numerical Study of Fundamental Fluid Dynamics using Navier-Stokes Equations

Geojoe Kuruvila, Principal Investigator
NASA Langley Research Center

Research Objective

To study vortex breakdown and to develop the ability to predict it.

Approach

The integral forms of the complete, unsteady, compressible, three-dimensional Navier-Stokes equations in the conservation form, cast in a generalized coordinate system, are solved numerically to simulate vortex breakdown. The inviscid fluxes are spatially discretized using Roe's upwind-biased flux-difference splitting scheme, and the viscous fluxes are discretized using central differencing. Time integration is performed using a backward Euler alternating-direction implicit scheme. A full approximation of multigrid is used to accelerate the convergence to steady state.

Accomplishment Description

A three-dimensional finite-volume Navier-Stokes code was developed to study vortex breakdown. The bubble-type breakdown was simulated for several sets of Reynolds numbers (Re), swirl velocity parameters (S), and low Mach numbers (M). Solutions were obtained for $S = 1$, $M = 0.1$ and Reynolds numbers, based on the vortex core radius, as high as 1000. A typical run with $65 \times 33 \times 33$ mesh points and four

multigrid levels required about 9 megawords of memory and 6 Cray-2 hours to obtain a solution converged to machine zero. About 2500 multigrid cycles were required to obtain a solution.

Significance

The code that has been developed is robust, and its performance is good for this particular problem, in terms of both the rate of convergence and the use of computer resources. The code is a good platform from which to conduct further extensive numerical experiments. Since the compressible form of the Navier-Stokes equations is used, this code can be used to study the effect of Mach number, in addition to the effects of Reynolds number and swirl velocity parameter, on the breakdown.

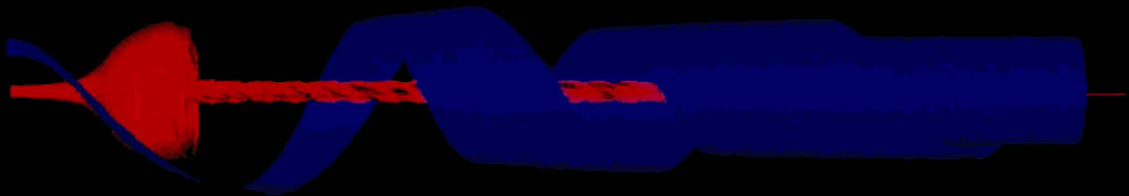
Future Plans

Other types of breakdown, especially the spiral type, will be studied, as will the effects of boundary conditions on the type of breakdown.

Publications

A NASA Technical Memorandum detailing this work will be published soon.

VORTEX BREAKDOWN



Streamsurfaces ($Re=375$, $S=1.46$, $M=0.1$)

Incompressible Navier-Stokes Calculations for the Space Shuttle Main Engine

Dochan Kwak, Principal Investigator

Co-investigators: Stuart Rogers, Seokkwan Yoon, Moshe Rosenfeld, and Cetin Kiris

NASA Ames Research Center

Research Objective

To develop numerical simulation procedures for viscous incompressible flows and apply them to the Space Shuttle main engine (SSME), high-lift aerodynamic configurations, and an artificial heart.

Approach

The three-dimensional, incompressible Navier-Stokes equations in generalized coordinates are solved by two different methods (pseudocompressibility and a fractional-step method) using different numerical algorithms.

Accomplishment Description

One of the primary areas in which the performance of the SSME can be improved is the flow in the high-energy liquid-oxygen and fuel pumps. The INS3D-LU code, which is based on the lower-upper, symmetric, Gauss-Seidel implicit algorithm, was used to study the effect of tip clearance in the inducer of the SSME high-pressure oxidizer turbopump. The time-accurate incompressible code, INS3D-UP, was used to study the unsteady flow during the opening and closing of a tilting disk valve. This is an idealized problem for an artificial heart valve, although the same code can be used to simulate the start-up or the shut-down procedure of the SSME. The total

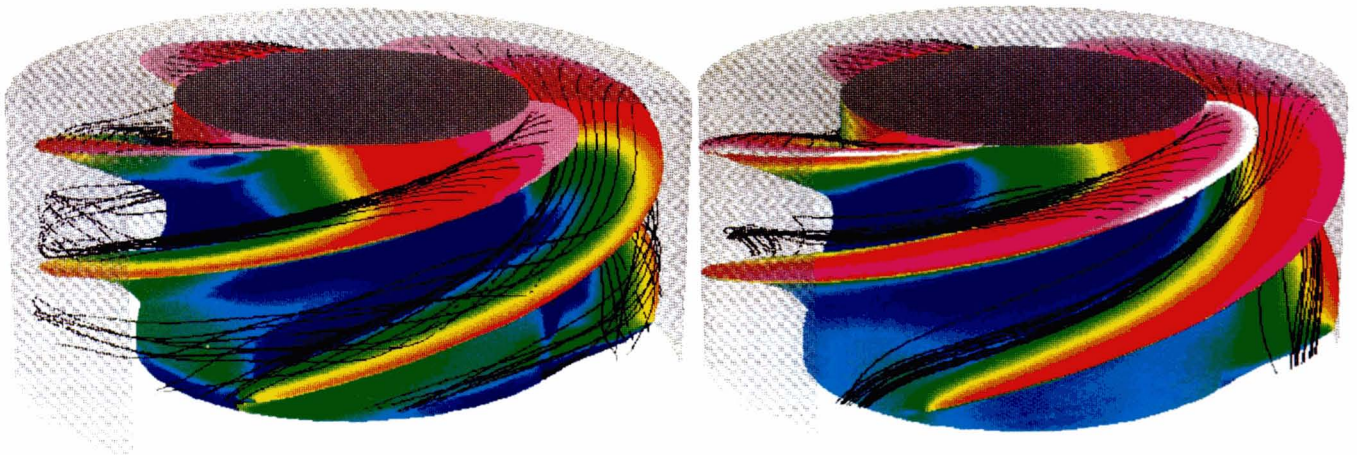
allocated time of 300 hours was used, and over 200 deferred hours were also used. An average run for the SSME inducer was kept small during this period, requiring approximately 15 megawords and 1 hour of run time. A finer resolution study and a more complete inducer/impeller geometry will require at least 3 to 5 times more mesh points and run time. An unsteady case of valve flow requires 5 megawords of memory and 10 hours of run time per cycle, using 100,000 mesh points.

Significance

The speed and accuracy of the incompressible Navier-Stokes codes developed are demonstrated by computing the SSME inducer and a tilting disk valve. By multi-tasking the INS3D-LU code on eight processors of the Cray Y-MP, 1.07 GFLOPs was achieved using 1 million grid points. This is a major step toward developing a highly efficient computational fluid dynamics design procedure.

Future Plans

A full simulation of the entire SSME power head will be continued. The numerical simulation of high-lift aerodynamic configurations will be performed. During the next period, an average job will require 10 to 20 megawords of memory and 10 to 40 hours of Cray-2 time.



The Space Shuttle main-engine turbopump (left) with tip clearance, and (right) without tip clearance.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Computation of Single-Expansion-Ramp and Scramjet Nozzles

H. T. Lai, Principal Investigator
Sverdrup Technology, Inc.

Research Objective

To obtain three-dimensional numerical flow fields for the nonaxisymmetric single-expansion-ramp and scramjet nozzles at test conditions in the supersonic regimes.

Approach

The existing PARC code is used to compute the solutions of laminar and turbulent thin-layer Navier-Stokes equations for a perfect gas.

Accomplishment Description

Converged turbulent results for the single-expansion-ramp nozzle were computed for a nozzle pressure ratio of 4. In this case, the flow at a stagnation reservoir expands subsonically along the converging section, becomes sonic at the throat and supersonic in the diverging section, and then exhausts supersonically into a quiescent environment. The internal expansion and the external exhaust plume were both simulated. A grid of

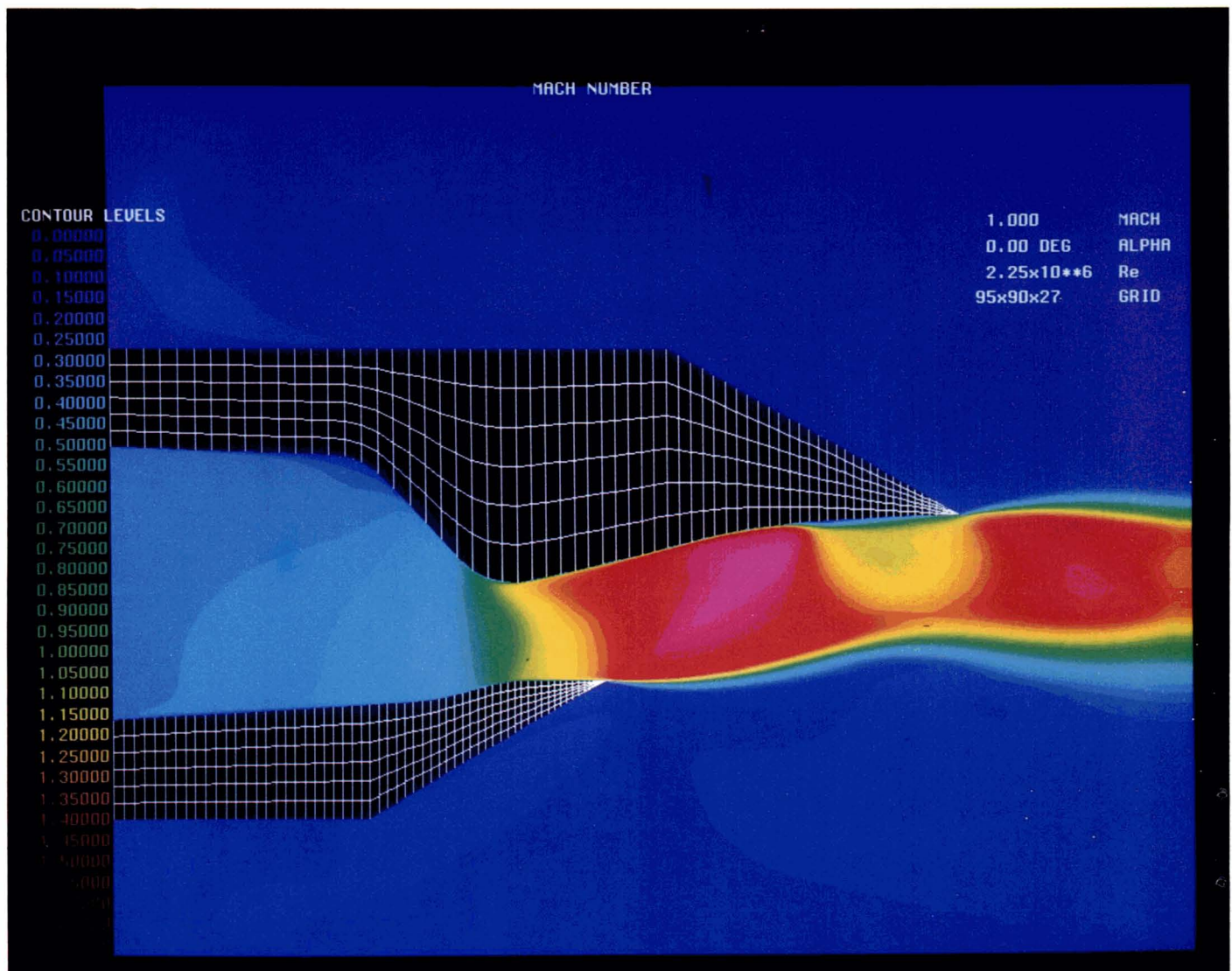
$95 \times 90 \times 50$ and a total of 45 Cray-2 hours were used. Converged laminar results for the scramjet nozzle were also obtained for an external Mach number of 6 and an internal Mach number of 1.62. The flow field is characterized by strong supersonic expansions in both streamwise and spanwise directions. Only a portion of the exhaust plume was simulated in this case. A grid of $90 \times 90 \times 95$ and a total of 18 Cray-2 hours were used.

Significance

Streamwise vortices are present in the exhaust flow of the single-expansion-ramp nozzle and on the ramp surface of the scramjet nozzle.

Publications

Lai, H. T. "3D Computation of Single-Expansion-Ramp and Scramjet Nozzles." Presented at the CFD Symposium on Aeropropulsion, Cleveland, OH, Apr. 1990.



Flow-field calculation; $M = 1.000$, $\alpha = 0.00^\circ$, $Re = 2.25 \times 10^6$, grid size $95 \times 90 \times 27$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Numerical Investigation of NASP-Like Vehicles

C. C. Lee, Principal Investigator

Co-investigators: William Bower, Shawn Hagmeier, Scott Van Horn, Pat Vogel, and Darrell Weber
McDonnell Douglas Corporation

Research Objective

The objective of this research is to calibrate National Aero-Space Plane (NASP) -related computational fluid dynamics (CFD) codes using the most recent test data from the McDonnell Douglas Corporation (MDC) -NASP project and the NASP Technology Maturation Plan (TMP). The calibrated CFD codes will then be used to produce design guidelines for NASP-like vehicles.

Approach

Three-dimensional Navier-Stokes codes developed by government agencies were selected for code calibration, including the CFL3DE and CNS codes. Reliable experimental data were extracted from the MDC-NASP project and the TMP. The code calibration included a grid density study, conservation error checks, and evaluation of physical modeling.

Accomplishment Description

In order to increase confidence in the CFL3DE code, an extensive calibration for flow-field and force/moment predictions was performed. It included calibrations against detailed flow-field data from wind tunnel and flight tests over a range of Mach numbers from 6 to 20. Configurations selected for this study included blended wing-body, noncircular-body, reentry F, advanced manned interceptor, and NASA Ames all-body. A

typical example of the computed flow fields is shown in the figure, which is for the all-body vehicle with Mach = 7.4. For this case, 49 radial, 49 circumferential, and 72 axial points were used. It was run in parabolized Navier-Stokes mode and took 25 CPU minutes on the Cray-2. Computed pitot pressure showed excellent agreement with the test data. Surface pressure and heat transfer rates showed the same agreement.

Significance

The CFD calculations are needed to predict flow fields for the NASP design in flight regimes beyond those in which current wind tunnels can operate. The accuracy of these predictions must be quantified using available data. All sources of errors present in the computation methods, post-processing methods, and experimental data must be quantified to reduce design risk. The calibrated CFD codes are necessary tools for the NASP design.

Future Plans

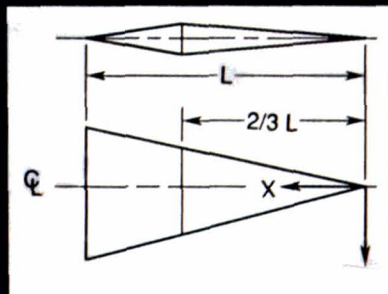
Code calibration for external and internal flows will continue. Grid density studies will be performed with all codes at different Mach ranges, to obtain grid-independent solutions. The calibrated codes will be used to perform parametric studies to define the critical geometric parameters that drive vehicle design.

MDC-CFL3DE Flowfield Solution for NASA Ames All-Body Hypersonic Vehicle

Mach 7.4

$\alpha = 0.0$

Single Zone Grid
(72x49x49)



The MDC-CFL3DE flow-field solution for the NASA Ames all-body hypersonic vehicle; $M = 7.4$, $\alpha = 0.0^\circ$, single-zone grid size $72 \times 49 \times 49$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Nonlinear Instability of Shear Layers

S. J. Leib, Principal Investigator
Sverdrup Technology, Inc./NASA Lewis Research Center

Research Objective

To study the nonlinear spatial evolution of instability waves on compressible shear flows.

Approach

A combination of asymptotic and numerical methods is used. Nonlinear effects first become important in the "critical layer" when the wave amplitude becomes large enough while the dynamics outside this region are still linear. An asymptotic analysis yields a nonlinear integro-differential equation for the instability wave amplitude, with the linear solution as an initial condition. The cubic nonlinearity has the form of a convolution integral over the entire history of the solution.

Accomplishment Description

A code was written to obtain numerical solutions to this nonlinear evolution equation. A fourth-order predictor-corrector scheme was used to advance the solution in the downstream direction. The trapezoidal rule was used to compute the integral term, and the upstream tails were evaluated analytically from the linear solution. Typical runs required from 4 to 6 megawords of memory and approximately 2 hours of CPU time.

Significance

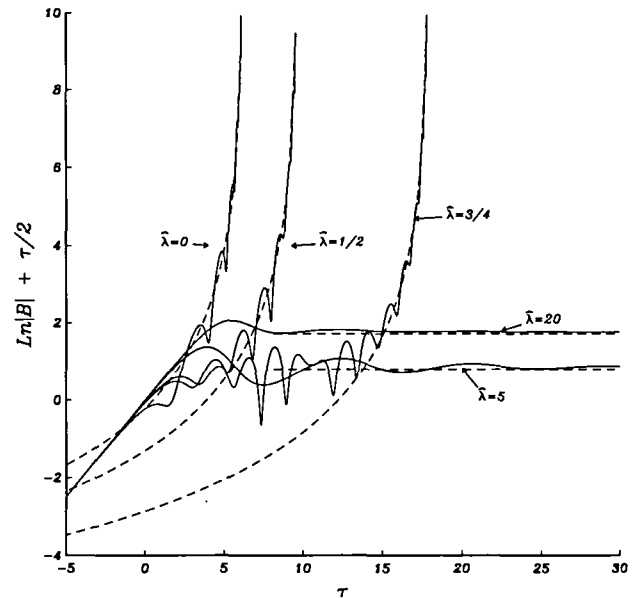
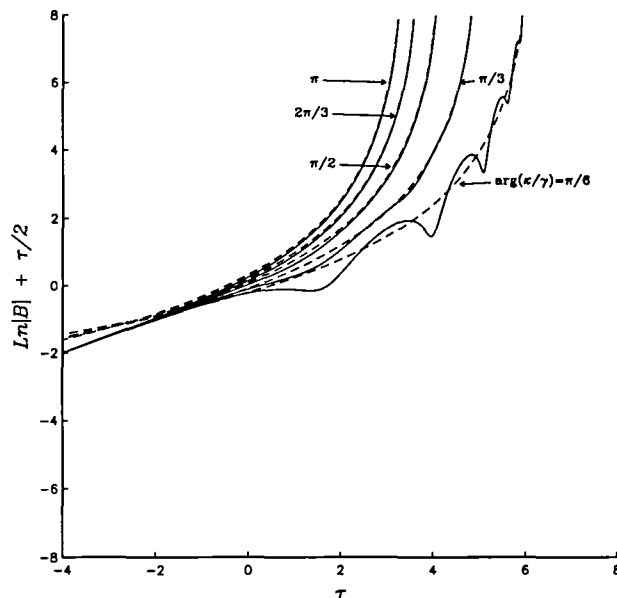
The numerical results showed that inviscid solutions ended in a singularity at a finite downstream distance. Equilibrium solutions were possible when the viscous effects were large enough, but only for certain parameter ranges. These results indicate that nonlinearity can lead to explosive growth of the instability wave for a compressible shear layer, in contrast to the incompressible case.

Future Plans

Similar evolution equations can be derived for other types of shear flows, and their numerical solutions will be pursued in the future.

Publications

1. Goldstein, M. E., and Leib, S. J. "Nonlinear Evolution of Oblique Waves on Compressible Free Shear Layers." *J. Fluid Mechanics* 207 (1989): 73-96.
2. Leib, S. J., and Goldstein, M. E. "Nonlinear Evolution of 3D Instability Waves on Compressible Shear Layers." Presented at the American Physical Society Division of Fluid Dynamics Meeting, Palo Alto, CA, 1989.
3. Leib, S. J. "Nonlinear Evolution of Subsonic and Supersonic Disturbances on a Compressible Free Shear Layer." Submitted to *J. Fluid Mechanics*, 1990.



Normalized instability wave amplitude vs. scaled downstream distance. Solid lines are numerical solutions, dashed lines are asymptotic. (Left) Inviscid solutions. (Right) Solutions for various values of the viscous parameter.

Computational Fluid Dynamics Analysis of Space Shuttle Main Engine Turbopump Rotor/Stator Flows in Two and Three Dimensions

S. J. Lin, Principal Investigator

Co-investigator: R. J. Yang

Rockwell International, Rocketdyne Division

Research Objective

The objective of the present research is to utilize and extend the rotor family codes developed by Dr. M. M. Rai at NASA Ames to calculate dynamic loading on various Space Shuttle main engine (SSME) turbine blades. Extensive flow data were obtained and then reduced to provide inputs for the dynamic and structural analysis of the turbine blade cracking problem in the SSME.

Approach

The rotor codes developed by Dr. Rai were applied in this project. The code was also extended to have multistage capability. The code solves the two- and three-dimensional, compressible, unsteady, thin-layer Navier-Stokes equations on a system of patched and overlaid grids.

Accomplishment Description

The Rotor-2 code developed by Dr. Rai was extended to have multistage capability. The code was also modified to compute steady-state subsonic and supersonic isolated airfoils. The resulting code was used to compute dynamic loading on turbine blades for a single-stage and a multistage high-pressure fuel turbopump, a multistage low-pressure oxidizer turbopump, and a single-stage single-crystal turbine blade. Good agreement was obtained between calculated results and SSME balance data. Extensive data were also obtained

and input to dynamic and structural codes for fatigue crack analysis. The calculations took about 4 to 30 Cray-2 hours for a single run, depending on the number of stages included. It required about 2 to 15 megawords of run time memory.

Significance

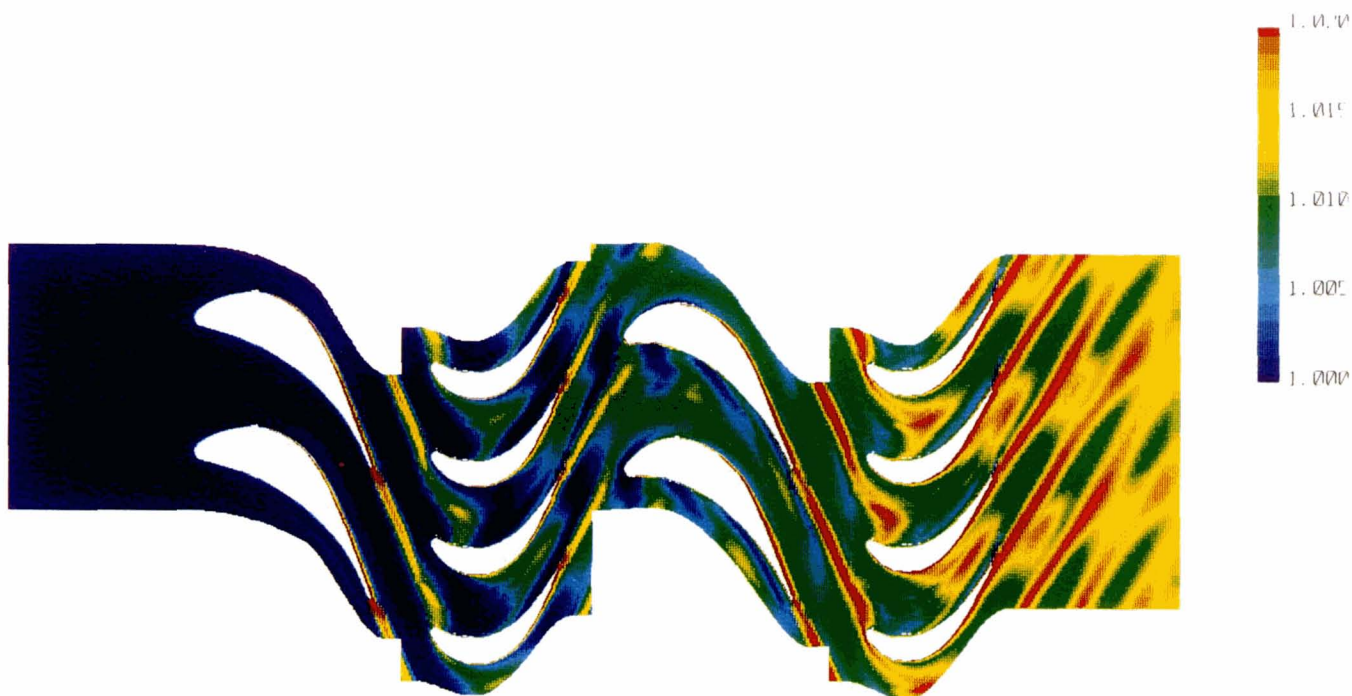
Turbopump blade cracks have been identified in the SSME. The life of these turbine blades is limited by high cycle fatigue cracks. Successful simulations of rotor/stator flows will provide accurate predictions of rotor/stator blade dynamic loading, which will help improve the performance and design of turbopumps.

Future Plans

The rotor code will be further developed to calculate dynamic loading and thermal characteristics of turbine blades in two and three dimensions. The data will then be used in dynamic and stress analysis codes for turbine crack analysis.

Publications

1. "Navier-Stokes Flow Simulations of Multi-Stage and Multi-Blade Turbines in SSME Turbopumps." Presented at the 8th CFD Working Group Meeting, Marshall Space Flight Center, AL, Apr. 1990.
2. "Rotor Code Applications to Turbine Analysis." Presented at the 8th CFD Working Group Meeting, Marshall Space Flight Center, AL, Apr. 1990.



Entropy contours for a two-stage, multistator/multirotor, high-pressure fuel turbopump.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Application of Computational Fluid Dynamics Methods to Radar Cross Section Computations

Rung T. Ling, Principal Investigator
Northrop Aircraft Division

Research Objective

The long-term objective of this project is to provide an integrated approach to solving aerodynamics and electro-magnetics problems in aircraft design. The short-term objective is to apply computational fluid dynamics (CFD) methods to radar cross section (RCS) computations.

Approach

A differential-equation approach was developed for the solution of the Maxwell/Helmholtz equations encountered in electromagnetic scattering. This approach is based on mathematical similarities between aerodynamics and electro-magnetics problems. It takes advantage of progress made in CFD and incorporates appropriate innovative modifications necessary for electromagnetics problems.

Accomplishment Description

A time-dependent method for the numerical solution of the frequency-domain Helmholtz equation was developed. This method involves the reversion of the Helmholtz equation to the wave equation, and the introduction of time dependence into the generalized scattering amplitude. The scattering problem thus reverts from an elliptic boundary-value problem to a hyperbolic initial-value problem. The hyperbolic nature of the wave equation allows a time-dependent method to be incorporated so that solutions can be obtained by forward marching in time. This time-dependent formulation has been applied to scattering by long cylinders and spheres. The accompanying figure shows the evolution history of the profiles of the time-dependent generalized scattering amplitude in

the scattering by a metallic circular cylinder. An explicit finite-difference scheme based on the Lax-Wendroff method is now being used in the three-dimensional RCS prediction code. The RCS computation for a three-dimensional object approximately one wavelength in each dimension needs 5 Cray-2 hours and 20 megawords of memory.

Significance

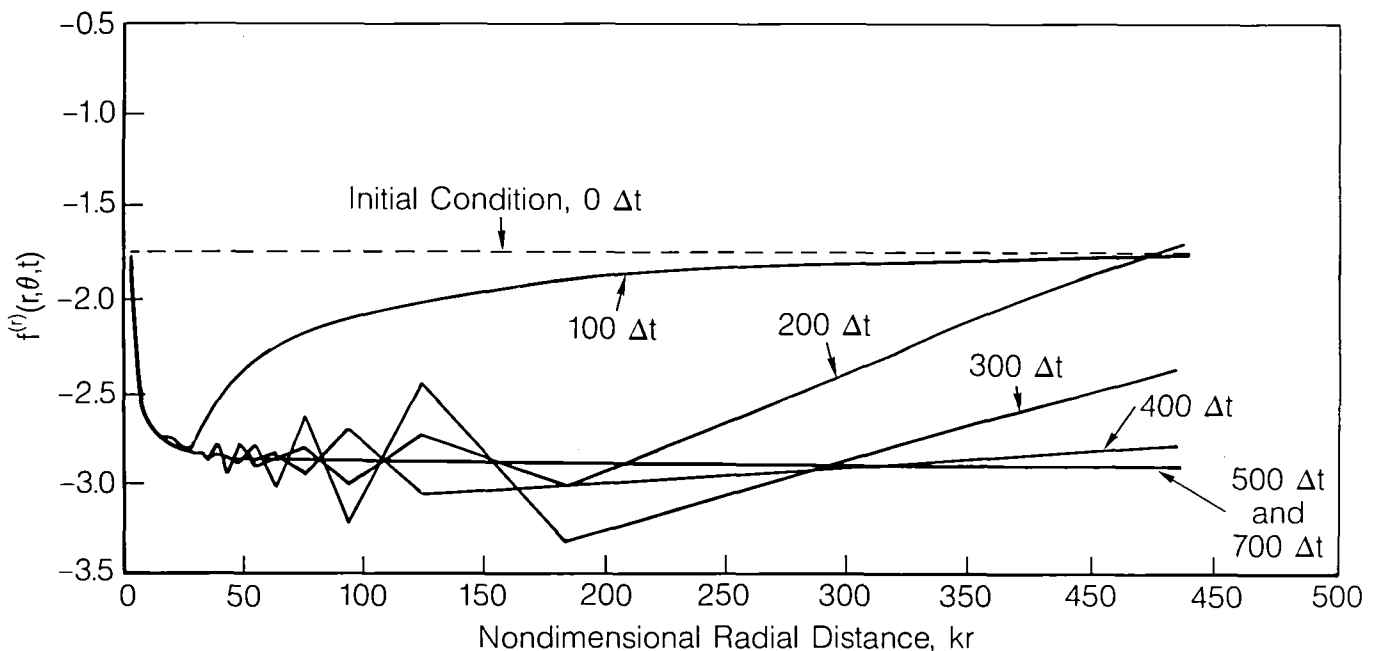
The time-marching method developed under this project for the solution of the frequency-domain Helmholtz equation is similar to using time-marching schemes to solve time-dependent fluid dynamic equations as a means of obtaining the steady-state solution in steady flow problems. The memory requirement is minimized by using an explicit time-marching scheme to allow three-dimensional electromagnetic scattering phenomena in the resonance and high-frequency regions to be accurately simulated.

Future Plans

The three-dimensional computational electromagnetics code will be further developed, validated, and applied to simple, nonspherical, three-dimensional objects.

Publications

"A Time-Dependent Method for the Numerical Solution of Wave Equations in Electromagnetic Scattering Problems." Invited paper, presented at the IEEE/URSI Meeting, San Jose, CA, June 1989. (Also to be published in a special issue of *Computer Physics Communications* on computational electromagnetics.)



Scattering by a metallic circular cylinder with $ka = 3.1$ and E-polarization; the time-dependent generalized scattering amplitude at various time steps ($\theta = 0$, real part).

Computational Prediction and Control of Steady and Unsteady Asymmetric Vortex Flows around Cones

C. H. Liu, Principal Investigator
Co-investigator: Osama A. Kandil
NASA Langley Research Center/Old Dominion University

Research Objective

The major objective of the present research is to simulate, analyze, and control steady and unsteady asymmetric vortex flows around cones. The influential flow asymmetry parameters, such as the relative incidence, Mach number, Reynolds number, and shape of cross-sectional area, are considered in this work. Control of flow asymmetry is also studied.

Approach

The unsteady, compressible Navier-Stokes equations were solved in this study. Two computational schemes were used to solve the equations. The first, which was the main scheme, was an implicit, upwind, flux-difference splitting finite-volume scheme. The second, which was used to validate certain solutions of the upwind scheme, was an implicit, approximately factored, central-difference finite-volume scheme. Pseudo-time stepping was used for unsteady-flow solutions. Locally conical flow solutions were considered at a chord station of unity.

Accomplishment Description

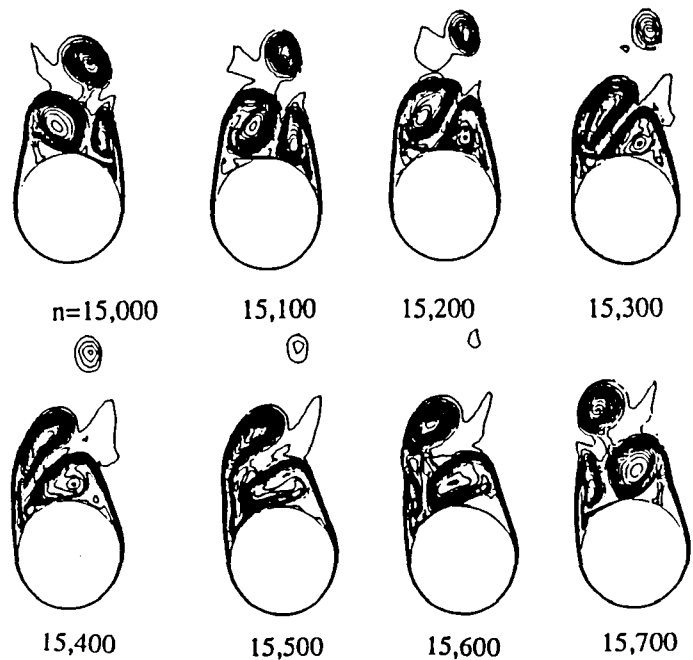
The figure shows sample solutions for unsteady, asymmetric, vortex-shedding flow. A grid of 161×81 points in the circumferential and normal directions was used for this application. The cone semi-apex was 5° . A modified Joukowski transformation with a clustering geometric series was used to generate the grid, and the smallest grid length at the solid boundary was 10^{-4} . The figure shows snapshots of the time history of the solution for the total-pressure loss contours. The time step from 15,000 to 15,700 represents one-half of the cycle of the periodic flow response. At the time step of 15,000, the asymmetric flow is seen with the shed vortex on the right side. As time progresses, the shed vortex is convected in the flow field and the primary vortex on the left side stretches upward, while the primary vortex on the right gets stronger and expands to the left side of the free-shear layer. At the time step of 15,600, the left side of the free-shear layer shows two vortices; one at the top and another one below it. The upper part of the free-shear layer is moving upward and is about to be shed in the flow field. The solutions at time steps 15,000 and 15,700 are mirror images of each other, and the shedding frequency is 4.49.

Significance

Results from the current investigation demonstrate the ability to simulate steady and unsteady asymmetric vortical flows around symmetric cones at zero side slip. This research shows, for the first time, that unsteady asymmetric vortex shedding is predicted.

Future Plans

Three-dimensional asymmetric flows are being considered for analysis and control. Several turbulence models are also being considered in this investigation to simulate other mechanisms of flow asymmetry.



Images of unsteady asymmetric vortex shedding; $M_\infty = 1.8$, $\alpha = 30^\circ$, $Re = 10^5$, $\Delta t = 10^{-3}$.

Transonic Navier-Stokes Solutions about a Complex High-Speed Accelerator Configuration

James M. Luckring, Principal Investigator

Co-investigators: Farhad Ghaffari, James L. Thomas, and Brent L. Bates

NASA Langley Research Center

Research Objective

To apply advanced Navier-Stokes methods to complex National Aero-Space Plane (NASP) -type configurations at transonic speeds.

Approach

An accelerator configuration recently tested in the Langley Research Center (LaRC) 16-Foot Transonic Tunnel was selected for the study. This model was composed of a cone-cylinder-frustum body, a wrap-around engine nacelle, forebody and aft-body engine fillets, and a 70° delta wing at incidence. The entire configuration surface was modeled analytically; this included faired-over engine inlet and exhaust faces. A blocked flow-field domain of approximately 373,000 grid points was then constructed with hybrid topologies, using transfinite interpolation methods. Steady-state solutions to the compressible, thin-layer Navier-Stokes equations were obtained with an implicit finite-volume algorithm developed at LaRC (CFL3D). These solutions were achieved using van Leer's upwind-biased, flux-vector-splitting technique and an extended version of the Baldwin-Lomax algebraic turbulence model.

Accomplishment Description

Turbulent results have been obtained at $\alpha = 2^\circ$, $R_L \approx 30 \times 10^6$ and $M_\infty = 0.9$, which correspond to flow conditions included in the recent wind tunnel tests mentioned above. The Mach

contours on the surface and in the plane of symmetry, shown in the accompanying figure, demonstrate a smooth solution across various longitudinally blocked interfaces. After a subsonic and mainly attached forebody flow, the flow accelerates supersonically over the engine cowl and subsequently shocks down at the exhaust cowl lip. (The sonic line is represented by a white contour line in the plane of symmetry to highlight the supersonic flow region.) The presence of the shock, as well as the slanted exhaust face, causes the flow to separate and envelop the entire boattail region. This separation appears to be the principal cause for the excessive number of iterations (4610) needed to achieve a three-order-of-magnitude reduction of the residuals. The solution required approximately 25 hours of Cray-2 time and 12 megawords of memory.

Significance

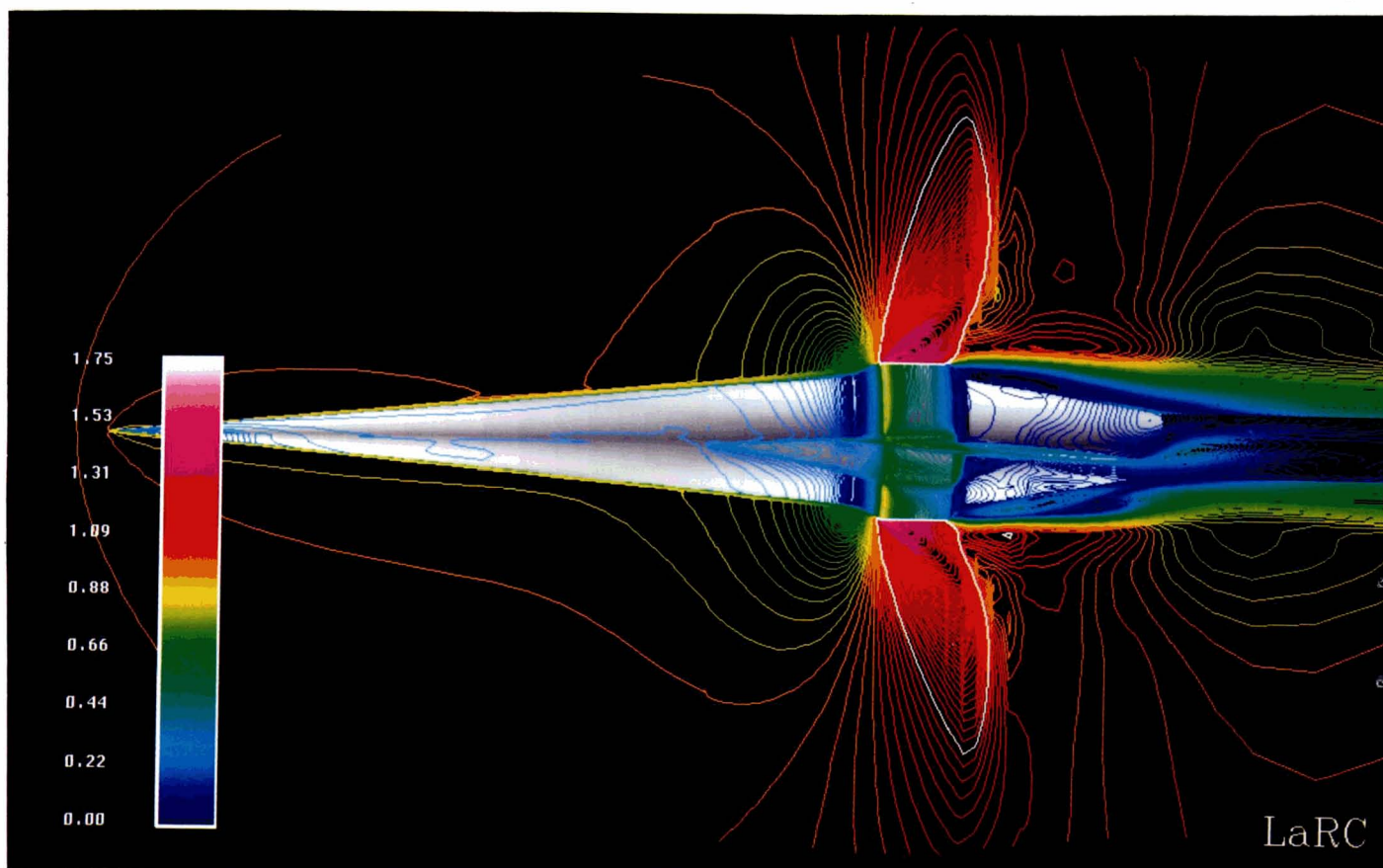
The design of proposed NASP vehicles will inevitably rely heavily on computational fluid dynamics to complement and extend information obtained in current ground-test facilities.

Future Plans

Efforts are under way to develop and implement a simple algorithm for simulating propulsion-induced effects.

Publications

Ghaffari, F.; Luckring, J. M.; Thomas, J. L.; and Bates, B. L. "Transonic Navier-Stokes About a Complex High-Speed Accelerator Configuration." AIAA Paper 90-0430, 1990.



Navier-Stokes Mach contours; $M_\infty = 0.9$, $\alpha = 2^\circ$, $R_L = 30 \times 10^6$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Earthquake-Cycle Stress Simulations

Gregory A. Lyzenga, Principal Investigator

Co-investigators: Arthur Raefsky and Stephanie G. Mulligan
Jet Propulsion Laboratory/California Institute of Technology

Research Objective

This work is aimed at understanding the spatial and temporal variations in stress and strain in the Earth's crust and upper mantle, at plate tectonic boundaries where relative motion is accommodated by recurrent great earthquakes. The resolution of outstanding questions regarding the driving forces of plate tectonics and seismic rupture requires a quantitative understanding of the processes operating in the transitional regime between elastic/brittle rock behavior and plastic/ductile flow.

Approach

Two-dimensional finite-element simulations of viscoplastic deformation were used to investigate the time evolution and distribution of stress near an infinitely long strike-slip fault in a layered Earth model. Special-purpose stress-activated fault elements were developed to simulate the stress-driven seismic rupture initiation.

Accomplishment Description

A systematic examination of a suite of candidate Earth models was conducted. The variable parameters included the far-field plate driving rate, the fault-failure strength criterion (either strong at ~ 100 MPa, or weak at ~ 10 MPa), and the crustal rheology law (either Maxwell-Newtonian or power-law non-Newtonian). The results of the study consist of calculated earthquake recurrence times, stress distributions, and stress/strain histories, which may be compared with available field observations from the San Andreas fault region. These model results include the prediction that the crustal stress field is characterized by a local minimum in shear stress at shallow depths centered on the fault trace. At greater depths, below

the elastic/plastic transition, the stress distribution is inverted, with locally high stress in the immediate neighborhood of the down-dip fault extension.

Significance

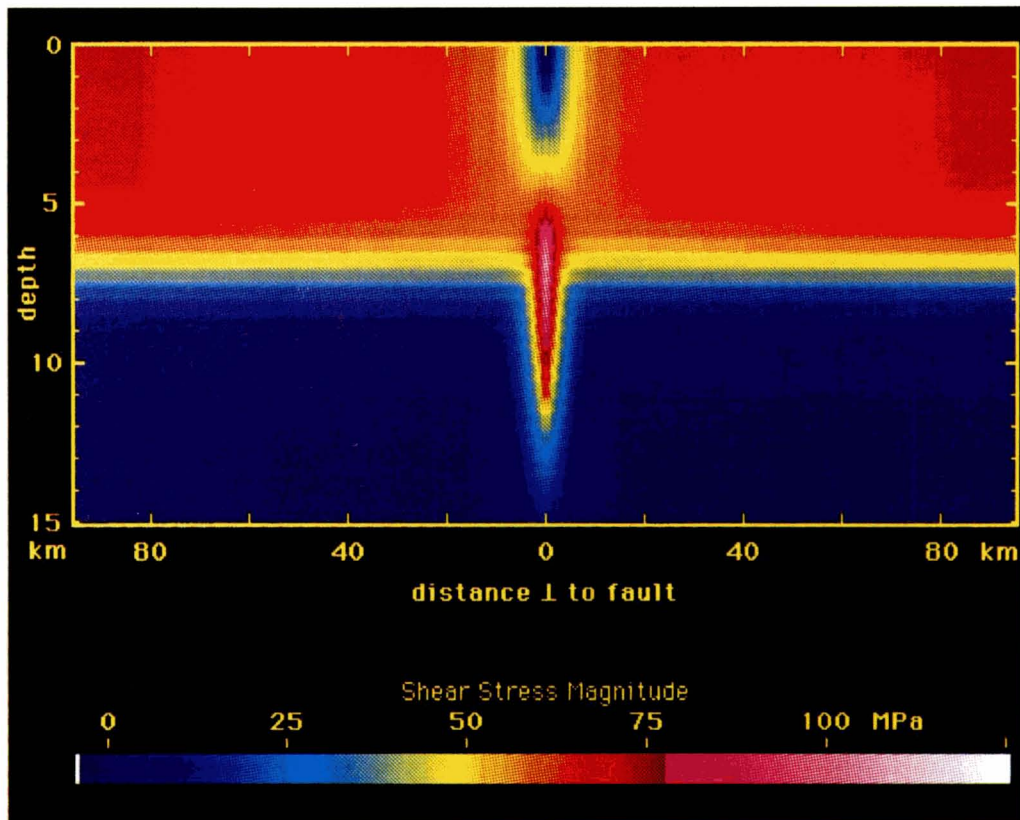
The results of simulations to date have lent support to the view that faults like the San Andreas fault generally sustain shear stresses an order of magnitude lower than those that would be expected on the basis of laboratory rock-strength measurements. The stress distribution features emerging from some of the current series of models are consistent with available experimental data, but also indicate areas in which additional measurements could be important. These results provide significant new details about the physical processes by which steady plate motion results from large episodic slip events on plate tectonic boundaries.

Future Plans

The existing finite-element code can be used to simulate deformations with three-dimensional displacement degrees of freedom on the two-dimensional grid. This capability will be used to simulate earthquake motions, including nonzero dip-slip components. Such models will be useful for the study of events like the October 1989 Loma Prieta, California, earthquake as well as convergent and divergent plate boundaries.

Publications

Lyzenga, Gregory A.; Raefsky, Arthur; and Mulligan, Stephanie G. "Models of Recurrent Strike-Slip Earthquake Cycles and the State of Crustal Stress." Submitted to *J. Geophysical Research*, Apr. 1990.



Plot of shear-stress total magnitude for the power-law non-Newtonian rheology model, high-stress fault case, showing the stress field at year 108 of the 220-year earthquake cycle. The stress distribution is qualitatively similar to that of the Newtonian case, except in the details of the brittle/ductile transition region.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Numerical Investigation of a Space Shuttle Orbiter Contingency Abort

Edward C. Ma, Principal Investigator

Co-investigator: Thomas C. Wey

Lockheed Engineering and Sciences Company

Research Objective

To investigate the contingency-abort longitudinal aerodynamic characteristics of the Space Shuttle orbiter and to compare the computational fluid dynamics results with experimental data.

Approach

A three-dimensional Euler code with a perfect-gas model is used to simulate high-angle-of-attack flow fields around the Space Shuttle orbiter.

Accomplishment Description

To integrate the Euler equations, three implicit methods are implemented; they are the two-factor alternating-direction implicit method, the blocked-line Gauss-Seidel scheme, and the hybrid time-marching scheme. A generalized upwind-biased interpolation scheme, which is basically an extension of van Leer's MUSCL scheme, was derived to achieve second- or third-order-accurate solutions. This new interpolation scheme was implemented in Steger-Warming and

van Leer's flux-vector splitting methods and also in Roe's flux-difference splitting method. As shown in the planform-view plot, the bow shock from the nose impinges on the wing near the side fuselage. As the angle of attack gets higher, the wing shock spreads outward and the interaction point moves off from the wing leading edge. The perfect-gas computation needed 2 Cray Y-MP hours and 3.9 megawords of memory.

Significance

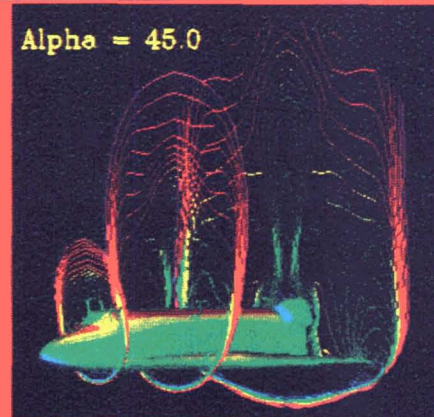
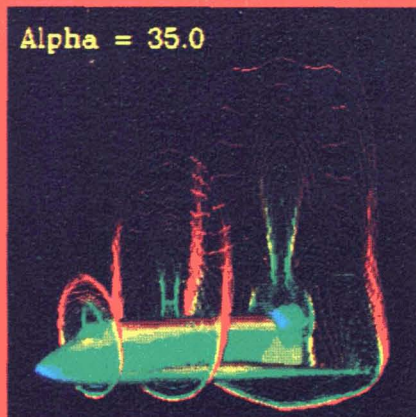
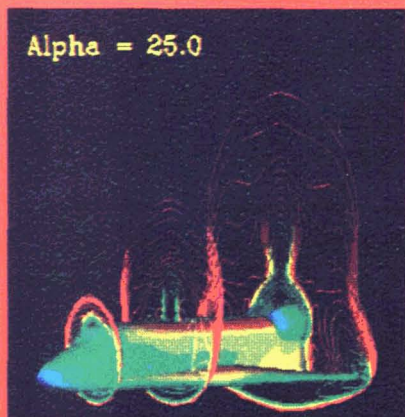
If a contingency abort is required during the ascent, the orbiter will have to fly at a very high angle of attack (45° or above) and a relatively low Mach number (between 1 and 4). Successful simulation of these high-angle-of-attack cases provides an indispensable tool in the decision-making during an abort entry.

Future Plans

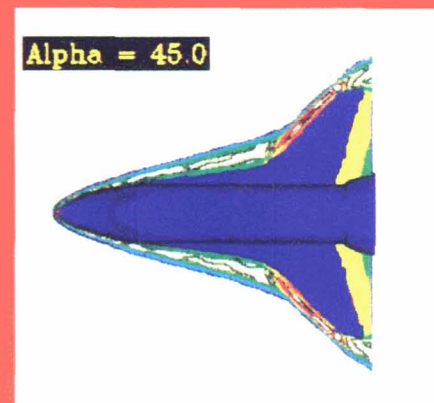
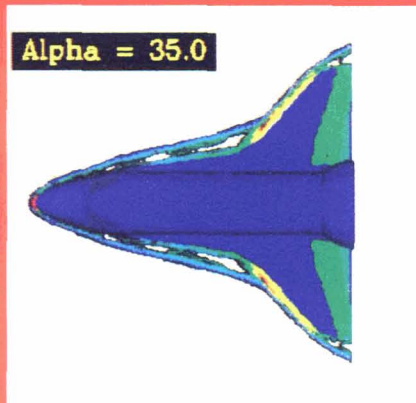
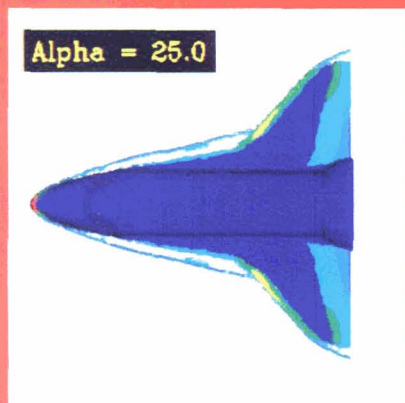
The codes will be further used to predict the aerodynamic characteristics of a complete Space Shuttle orbiter.

Shuttle Orbiter Mach Number Distribution

$M = 10.0$
80x35x65



Shuttle Orbiter Pressure Distribution



Space Shuttle orbiter Mach number (top), and pressure distributions (bottom); $M = 10.0$, grid size $80 \times 35 \times 65$.

Nonlinear Evolution of Supersonic Disturbances in Mixing Layers

M. G. Macaraeg, Principal Investigator
NASA Langley Research Center

Research Objective

To understand the nonlinear evolution of supersonic disturbances that dominate the instability processes of high-speed mixing layers.

Approach

Linearized and fully nonlinear compressible Navier-Stokes codes were written to study the linear stability and nonlinear evolution of high-speed mixing layers. The flow is considered to be temporally evolving and the solution schemes are a fully implicit spectral algorithm and an explicit algorithm, both written to study the nonlinear evolution.

Accomplishment Description

The fully implicit spectral algorithm uses a central finite-difference preconditioner. Discretization involves a Chebyshev series in the normal direction and a Fourier series in the streamwise direction. An order-of-magnitude increase in the allowable time step is attainable relative to an explicit formulation. Linear stability analysis identified the dominant instability modes associated with high-speed mixing layers to be supersonic. Studies of these disturbances in the nonlinear simulation show a resistance to subharmonic growth—a necessary precursor for mixing enhancement. Running the explicit code requires 3.66×10^{-5} seconds per time step per grid point and 15 megawords of memory for a typical grid. The implicit code is competitive in total time required to reach a desired time level. Further work is under way to speed up the finite-difference preconditioner, which presently requires the largest block of time in this code.

Significance

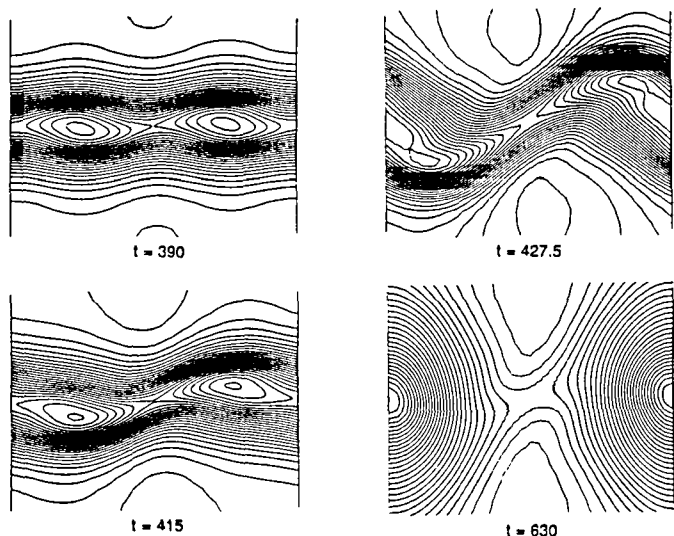
Enhanced mixing between the fuel stream and airstream is a major issue in scramjet engine design. Experiment and theory have shown that increases in Mach number produce lower levels of mixing efficiency. The current study shows how the growth of primary disturbances at low Mach number (left-hand figure) evolves into a strong subharmonic disturbance (i.e., vortex pairing)—a mechanism responsible for mixing enhancement. This situation is sharply contrasted with that in the right-hand figure, which shows the time evolution of a supersonic disturbance. No subharmonic growth occurs; instead, there is a spectral broadening, as evidenced by the development of small-scale structures.

Future Plans

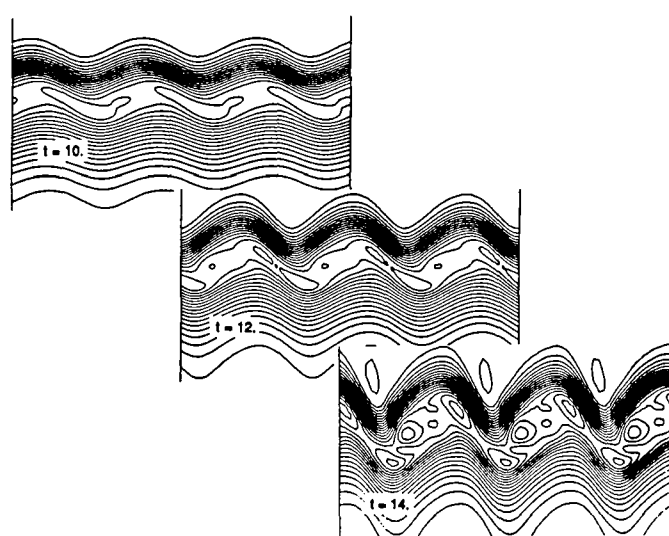
To study the complex nonlinear interactions of supersonic disturbances, secondary stability analysis will be used in conjunction with the nonlinear simulation. The secondary analysis will provide insight into the types of interactions these modes may undergo to promote higher harmonic growth and thus enhanced mixing. These studies will be pursued in the full nonlinear simulation study.

Publications

1. Macaraeg, M. G., and Streett, C. L. "New Instability Modes for Bounded, Free Shear Flows." *Physics of Fluids A* 1, no. 8 (Aug. 1989).
2. "Bounded Free Shear Flows: Linear and Nonlinear Growth." *Proceedings of the ICASE/NASA Langley Workshop on Stability and Transition*, June 1989.



Subharmonic growth of a Mach 0.5 free-shear flow shearing mode as time progresses.



Growth of a supersonic "kinking" mode; $M_\infty = 2$.

Simulation of Highly Ionized Flows

Robert W. MacCormack, Principal Investigator

Co-investigator: Philippe Rostand

Stanford University/Analatom, Inc.

Research Objective

The ultimate objective is to obtain the capability to predict the flow of a moderately ionized nitrogen plasma in the different components of an arc-heated wind tunnel. The techniques developed will also be useful for the design of electric plasma thrusters. As a first step, we derived a computational fluid dynamics program capable of simulating the recombination of the plasma in an expansion chamber.

Approach

A generic program for the simulation of an axisymmetric flow of nitrogen in thermochemical nonequilibrium at high enthalpy and high speed is used to investigate the behavior of the jets of interest. The electric effects are accounted for in the so-called weak ionization approximation. The numerical method chosen is a finite-volume, upwind implicit scheme.

Accomplishment Description

Our program was developed and its results compared with experiment to obtain preliminary validation. The program uses five species, three temperatures, and the two components of the speed vector to represent the media. The internal energy model includes vibration of molecules, electronic excitation of atoms, and thermal agitation of electrons. The chemical model consists of six two-body and three-body reactions; the influence of the thermal nonequilibrium on the rates of reaction is accounted for. The electric effects introduce a nonconservative term in the equations, which is the source of numerical

difficulties. The time integration to steady state is done through a fully coupled, fully implicit nonfactored scheme. A typical calculation takes 10 minutes on the Cray-2 and uses 3 megawords of memory. The accompanying figure shows the mass fraction of atomic nitrogen in the mixture, in the calculation that was used for validation. It illustrates the fact that the stiffness problem and its associated numerical difficulties have been overcome.

Significance

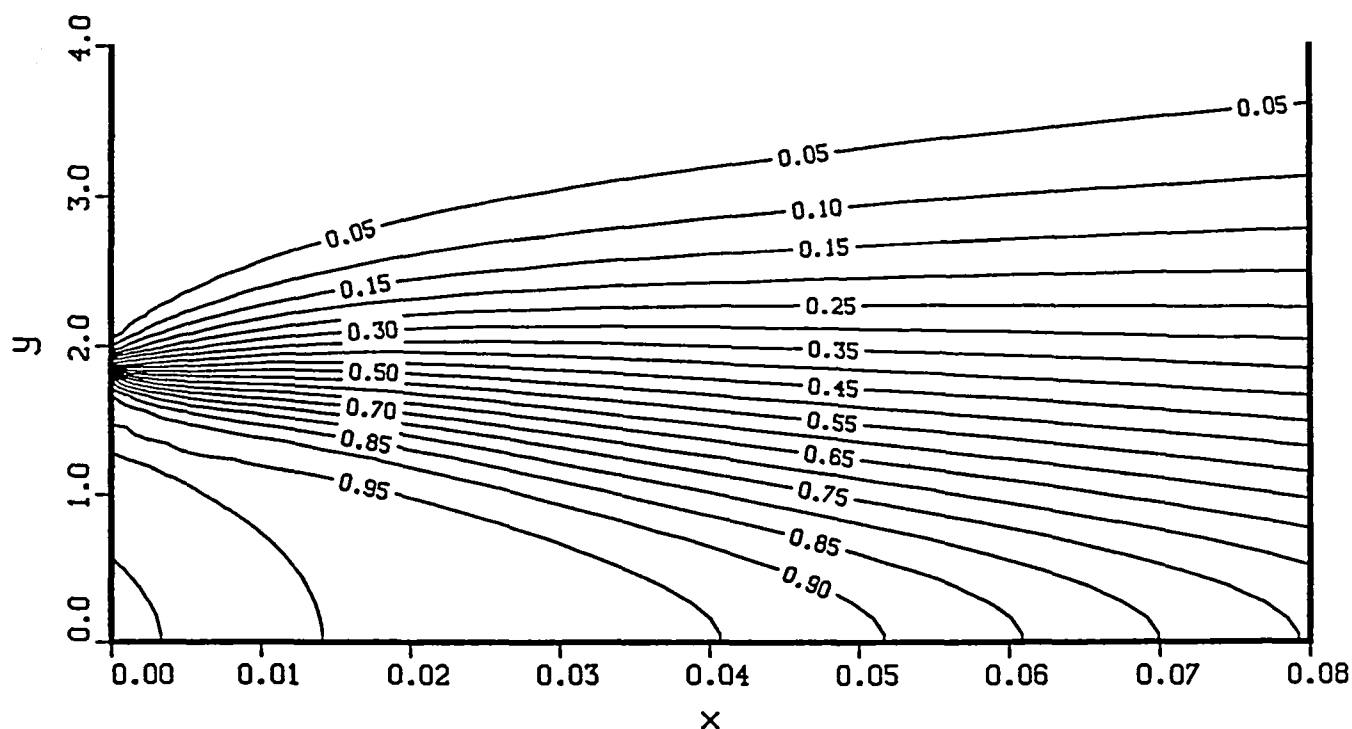
Arc-jets are one of the few means by which high-enthalpy flows can be investigated experimentally. Their numerical simulation is important for two related reasons: to help in their design, and to improve and validate the models of the numerous phenomena they host (dissociation, ionization, thermal nonequilibrium, and radiation are the main ones).

Future Plans

A more extensive validation of the code will have to include simulation of nozzle expansions, in which rapid cooling is usually a challenge to the thermochemical model. A natural extension of our work would be the simulation of the arc itself, which would need the inclusion of an electrical current model and a radiation model.

Publications

Rostand, P., and McCormack, R. W. "Nonequilibrium Flow in an Arc-Jet." Presented at the Workshop on Hypersonic Flows for Reentry Problems, Antibes, 1990.



Mass fraction of atomic nitrogen; grid size 120 × 65.

Computer Simulation of Ion Transport across Membranes

Robert D. MacElroy, Principal Investigator
Co-investigator: Andrew Pohorille
NASA Ames Research Center

Research Objective

The objective of this research is to provide a molecular-level description of protobiological processes. In particular, our goal is to explain phenomena occurring at interfaces between water and nonpolar media (such as a membrane interior), and to understand how ubiquitous biological processes could occur in the absence of the complex, specialized enzymes that mediate these processes in currently existing organisms.

Approach

Our approach is to simulate the behavior of all the atoms in the system by molecular dynamics (numerical integration of Newton's equations of motion). This method allows us to obtain a molecularly detailed description of the system and to calculate its thermodynamic and structural properties. The system consists of biochemically important molecules in their natural environment (water and/or membranes).

Accomplishment Description

(1) Interactions of positive (sodium) and negative (chloride and fluoride) ions with a water liquid/air interface were studied. It was shown that the water at the interface with nonpolar media is significantly different from that in the bulk. As a result, negative ions can approach the interface more easily than positive ions (30 CPU hours, 4 megawords of memory). (2) First simulations of molecules at water surfaces were performed. Amphiphilic molecules assume rigid orientations in which polar groups are buried in water while hydrocarbon parts are located in the nonpolar environment (20 CPU hours, 4 megawords of memory). (3) Large-scale simulations of DNA fragments in an aqueous solution were performed in order to study the opening of the DNA double helix. Two distinctly

different pathways of opening were found (50 CPU hours, 8 megawords of memory).

Significance

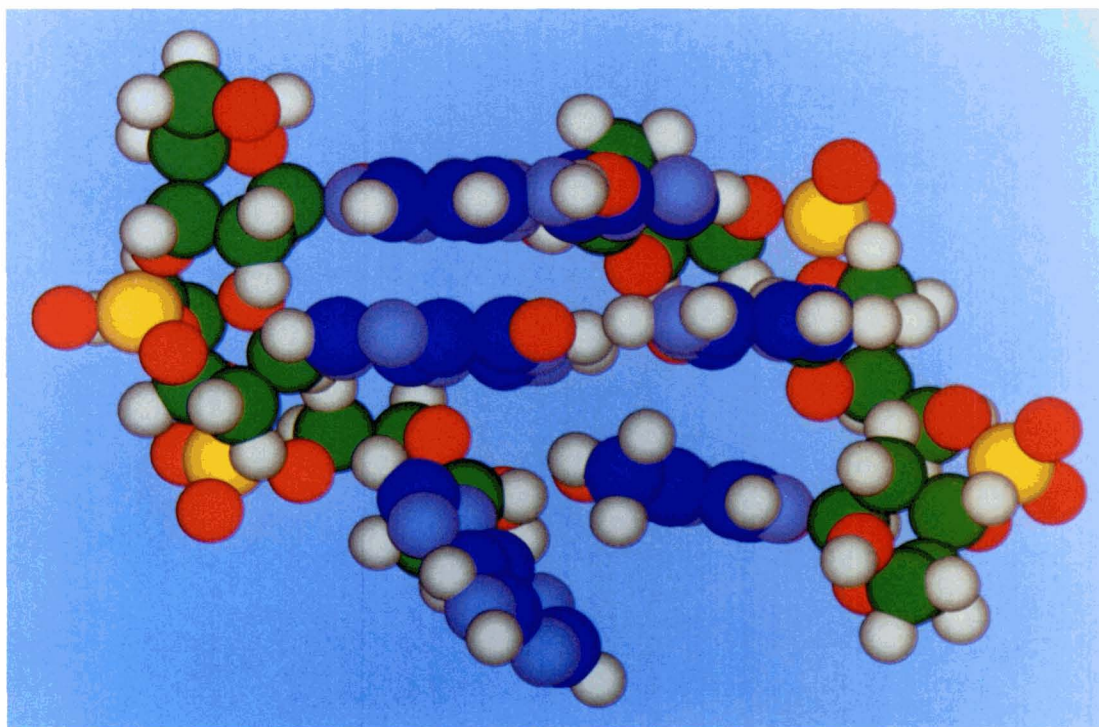
Our studies showed that interfaces between water and nonpolar media are uniquely suitable for promoting protobiological evolution. They can (1) selectively attract organic material and mediate its transport, (2) serve as simple catalysts for chemical reactions, and (3) promote the formation of electrical and chemical gradients, which could provide energy sources for protocells. Simulations of DNA yield information on the opening of the double helix—the critical step in protobiological, nonenzymatic DNA replication.

Future Plans

We will continue using computer simulations to study membrane/water interfaces, chemical reactions at those interfaces, and the transport of ions and small molecules across membranes.

Publications

1. "DNA Dynamics in Aqueous Solution: Opening the Double Helix." To be published in *Int. J. of Supercomp. Applications*, 1990.
2. Pohorille, A., and Pratt, L. R. "Cavities in Molecular Liquids and the Theory of Hydrophobic Interactions." *J. Am. Chem. Soc.*, 1990.
3. Wilson, M. A.; Pohorille, A.; and Pratt, L. R. "Comment on: Study on the Liquid-Vapor Interface in Water. I. Simulation Results of Thermodynamic Properties and Orientational Structure." *J. Chem. Phys.* 90 (1989): 5211.



A fragment of the DNA double helix with the terminal adenine-thymine base pair opened. For clarity, water molecules and counterions, present in the calculation, were removed. Oxygen atoms are red, nitrogen atoms are light blue, hydrogen atoms are white, phosphorus atoms are yellow, carbon atoms of the bases are dark blue, and carbon atoms of the sugar-phosphate backbone are green. Note that both hydrogen bonds between thymine and adenine are broken and adenine moved away from its original position in the double helix. In contrast, the position of thymine remains almost unchanged.

Transonic Analysis About the F-16A and Other Complex Configurations

Michael D. Madson, Principal Investigator

Co-investigators: Alex C. Woo, Ralph L. Carmichael, F. T. Johnson, S. S. Samant, D. P. Young, R. G. Melvin,

M. B. Bieterman, and J. E. Bussoletti

NASA Ames Research Center/Boeing Advanced Systems

Research Objective

To develop and validate a computational method that eliminates the use of surface-conforming grids in the analysis of complex aircraft configurations in the transonic flow regime.

Approach

The TranAir full-potential code was developed by Boeing under NASA contract. The code embeds accurately defined surface-panel models of very general and complex configurations such as the F-16A in a rectangular array of flow-field grid points. The grid is adaptively refined based on local error estimates, in order to more accurately resolve the local flow field. A system of nonlinear algebraic equations is constructed from the finite-element operators on the grid; the equations are solved in an iterative fashion.

Accomplishment Description

The final version of the code was delivered during the operational year. The two most significant enhancements to the code were a solution-adaptive grid-refinement capability and the ability to solve supersonic free-stream problems. The solution-adaptive capability helps the user determine where denser grid definitions are needed. The refinement in regions of secondary interest can be controlled by the user so that the grid-point distribution is clustered in regions of primary interest. The ability to solve problems in the supersonic flow regime allows engineers to obtain solutions for complex fighter configurations as well as for commercial high-speed transports. The figure shows a vertical cut of the refined grid through the wing and chine of the generic fighter geometry. This grid was

generated for a case in which $M = 1.2$ and $\alpha = 4^\circ$. The solution about a complex configuration in transonic flow using about 300,000 flow-field grid points takes about 5000 CPU seconds on a Cray Y-MP and uses less than 4 megawords of central memory.

Significance

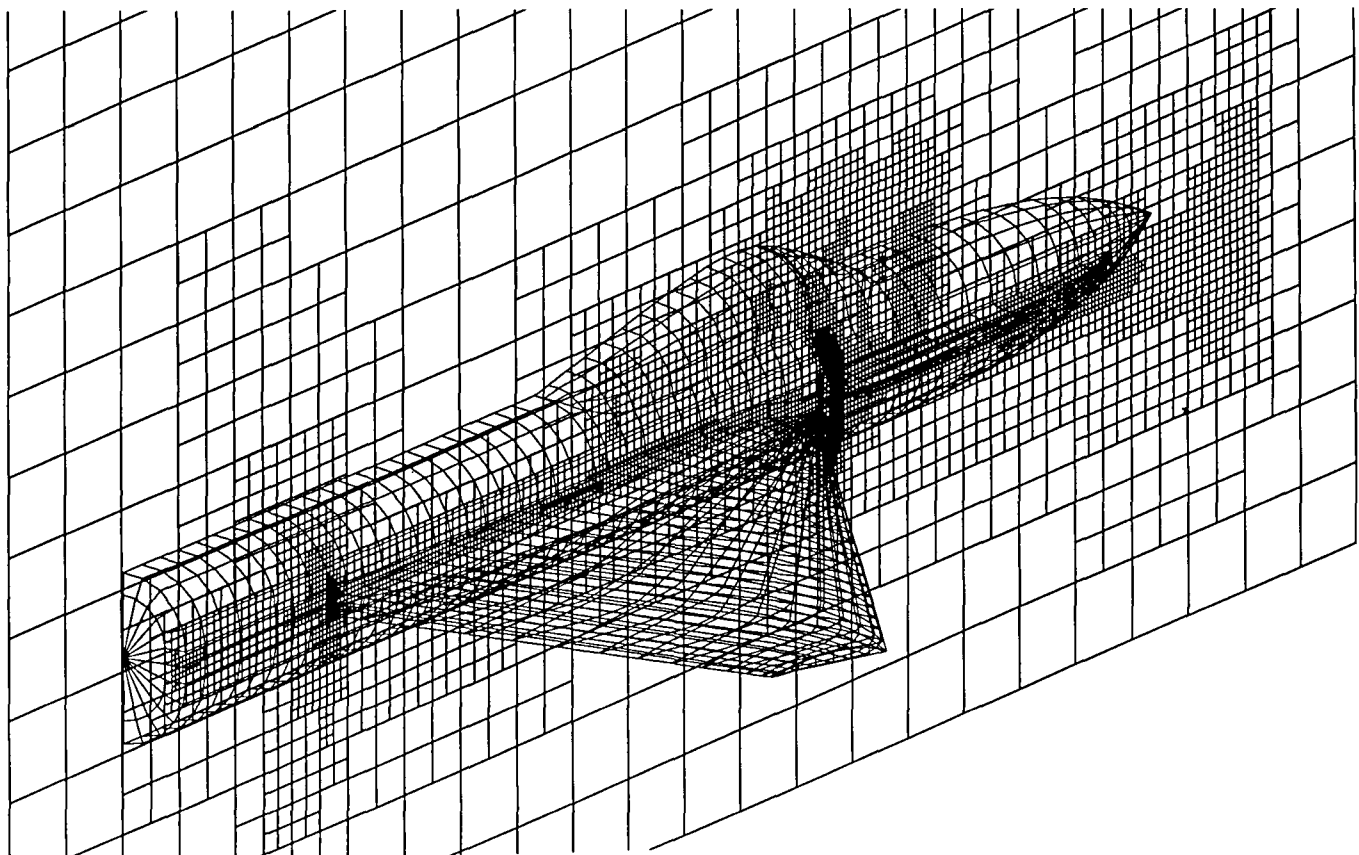
TranAir has the unique ability to obtain solutions about arbitrary and complex configurations in subsonic, transonic, and supersonic flow regimes. The use of surface panels for the geometry definition, and the use of unstructured Cartesian flow-field grids allow the engineer to quickly and easily obtain aerodynamic solutions for a wide range of configurations.

Future Plans

This project was not granted a continuation for operational year 1990-91. Work will continue on the ACF Cray Y-MP. The code is scheduled to be delivered to several requesters in government, industry, and academia.

Publications

1. Madson, M. D.; Carmichael, R. L.; and Mendoza, J. P. "Aerodynamic Analysis of Three Advanced Configurations Using the TranAir Full-Potential Code." NASA CP-3020, vol. 1, part 2, Apr. 1989, pp. 437-452.
2. Johnson, F. T.; Samant, S. S.; Bieterman, M. B.; Melvin, R. G.; Young, D. P.; Bussoletti, J. E.; and Madson, M. D. "Application of the TranAir Rectangular Grid Approach to the Aerodynamic Analysis of Complex Configurations." AGARD CP-464, May 1989, chap. 21.



A vertical cut of the refined grid through the wing and chine of a generic fighter geometry.

Radiation and Control of the Acoustic Field Caused by a Wave Packet Evolving along a Concave-Convex Surface

L. Maestrello, Principal Investigator

Co-investigator: N. M. El-Hady

NASA Langley Research Center/Analytical Services and Materials, Inc.

Research Objective

This research is an engineering application of flow through a wind tunnel contraction. The objective is to investigate the acoustic field and the control of the acoustic radiation resulting from the evolution of a wave packet in an unstable boundary layer over a concave-convex surface. Numerical experiments were conducted to study the behavior of a wave packet with regard to the stability, sound radiation, and control of the radiation in the boundary layer.

Approach

The analysis is based on the numerical solution of two-dimensional, nonlinear Navier-Stokes equations; a fourth-order-accurate operator-split algorithm is used. An initial steady solution is obtained, and an unsteady disturbance of finite duration is introduced in the form of a Tollmien-Schlichting wave and its subharmonic. Hence, a wave packet is created that grows and decays as it passes through the region of curvature. The acoustic radiation resulting from the evolution of the packet is evaluated by solving the linearized Euler equations, using the pressure computer from the Navier-Stokes equations as a time-dependent boundary condition. A fixed control algorithm is used in which the attenuated signal is

synthesized by a filter that actively adapts itself to the amplitude-time response of the outgoing acoustic waves.

Accomplishment Description

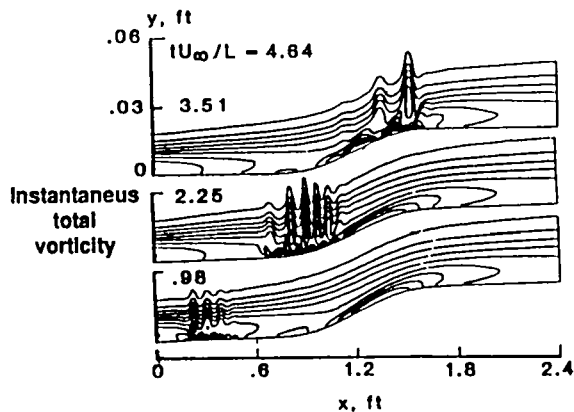
The character of the wave packet is illustrated by the instantaneous total vorticity, plotted in terms of nondimensional time. The total vorticity was initially amplified on the flat portion, and additional amplification comes on the concave curvature due to unfavorable pressure at 60° angle show a strong beaming, an equivalent weaker beaming occurs at 135°. The control is applied to the far field to adjust the amplitude of the interference signal at different angles. The filters compensate for the amplitude and bandwidths, and the controllers produce an interference field. The effectiveness of the control is shown by a reduction in acoustic intensity corresponding to a 6-dB change in amplitude.

Significance

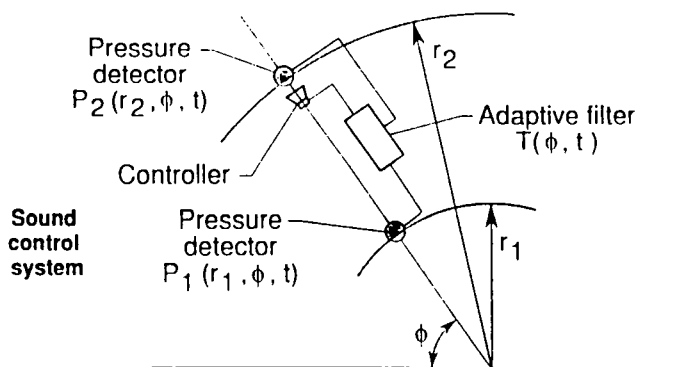
Inhomogeneity of the source field over a finite time has been identified as an important source of sound radiation and can be regarded as typical wind tunnel contraction noise.

Future Plans

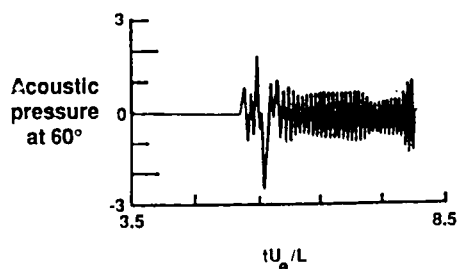
A model wind tunnel is being constructed using a new contraction geometry such that disturbances propagating into the contraction can effectively decay exponentially with distance.



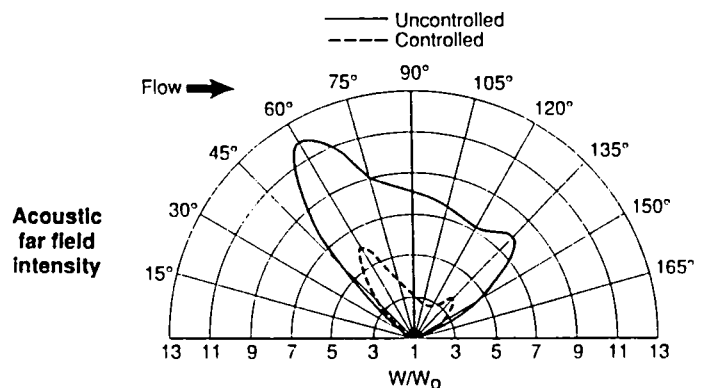
Instantaneous total vorticity.



Sound control system.



Acoustic pressure at 60°.



Acoustic far-field intensity.

Control of the acoustic field caused by a wave packet evolving over a concave-convex surface.

Transition in Hypersonic and Three-Dimensional Boundary Layers

Mujeeb R. Malik, Principal Investigator

Co-investigators: C.-L. Chang and R. E. Spall

High Technology Corporation/NASA Langley Research Center

Research Objective

The long-term goal of this research is to develop the capability to predict the onset of boundary layer transition in high-speed and three-dimensional boundary layers. Understanding the transition process in these flows is an essential prerequisite for the development of reliable transition-region models for computational fluid dynamics applications. This understanding and the prediction capability can only be achieved through studies involving internalization of disturbances (receptivity), linear and nonlinear development of various disturbances (Tollmien-Schlichting, crossflow, Görtler), and their possible interaction. For high-speed flows, such effects as bluntness, shock/wave interactions, and crossflow instability need to be studied.

Approach

For high-speed applications, Navier-Stokes and parabolized Navier-Stokes (PNS) codes are used for computation of the basic flow, which is analyzed using linear stability theory. Time-accurate Navier-Stokes codes are used for the direct numerical simulation of the nonlinear development of disturbances and final breakdown.

Accomplishment Description

(1) Linear stability analysis of the Mach 8 flow past a 7° semi-vertex blunt cone (nose Reynolds number $\approx 31,250$) was performed. Results indicated that nose bluntness caused the critical Reynolds number for the onset of instability to increase an order of magnitude (from about 2.25×10^5 for the sharp cone to 3.3×10^6 for the blunt cone). The predicted critical Reynolds number, the disturbance frequency, and the growth rates were in agreement with the experiment of Stetson et al. (1984). Calculations were also performed for reentry-F cone data ($M_\infty = 20$, 5° semi-vertex cone) using an equilibrium-gas model. These stability calculations required that the basic flow and its first and second derivatives be computed accurately.

About 240 grid points were used across the shock layer in the PNS calculation. A complete run (Navier-Stokes solution for the nose, PNS solution, stability analysis, and application of the e^N method) requires about 3 to 12 hours of Cray Y-MP time, depending on Mach number and gas chemistry. The maximum memory requirement is about 1.5 megawords for the axisymmetric version. (2) In order to understand the breakdown of a crossflow vortex (present in three-dimensional boundary layers because of inflectional crossflow velocity profiles), a Navier-Stokes simulation of a crossflow vortex in a three-dimensional boundary layer was performed. The resultant velocity field is shown in the figure. Because of the nonlinear crossflow vortex, low velocity regions appear within the boundary layer; this results in "wake-like" profiles subject to secondary instabilities. This would provide at least one route to transition in a three-dimensional boundary layer.

Significance

Transition prediction is crucial for the design of aerospace vehicles such as the National Aero-Space Plane, and for supersonic laminar flow control for High Speed Civil Transport. The agreement of our stability results for hypersonic flow with experimental data gives confidence in the theory and in its usefulness for transition prediction.

Future Plans

The codes will be used to study instabilities in three-dimensional boundary layers at supersonic and hypersonic Mach numbers for bodies at angle of attack. In addition, shock-wave effects on the instability will be studied and numerical simulations of transition will be performed.

Publications

"Effect of Nose Bluntness on Boundary Layer Stability and Transition." AIAA Paper 90-0112, 1990.

CONTOUR LEVELS

0.00000
0.05000
0.10000
0.15000
0.20000
0.25000
0.30000
0.35000
0.40000
0.45000
0.50000
0.55000
0.60000
0.65000
0.70000
0.75000
0.75000
0.80000
0.85000
0.90000
0.95000



Nonlinear crossflow vortex in a three-dimensional boundary layer. The computed velocity component along the axis of the vortex is shown.

Navier-Stokes Analysis of the Air Force Vortex Flap Model and the F/A-18

John Mangus, Principal Investigator

Co-investigators: Shreekant Agrawal, Robert Lowrie, and Brian Robinson

Northrop Aircraft Division/McDonnell Aircraft Company

Research Objective

The objective of this work is to investigate vortical flows about the Air Force Vortex Flap Model (AFVFM) and the F/A-18 aircraft at high angles of attack. The intent of these investigations is to analyze the actual geometries, to verify computational fluid dynamics codes for high-angle-of-attack flows, and to gain a better understanding of the physics of these flows.

Approach

Because of the complex geometries and the nature of the flow, three-dimensional Navier-Stokes codes were used to predict the flow fields. The thin-layer Navier Stokes code was used to compute the flow about the AFVFM geometry, and the CFL3D code was used to compute the flow about the F/A-18 Forebody-LEX geometry.

Accomplishment Description

Navier-Stokes solutions were obtained about the AFVFM and F/A-18 geometries. The AFVFM was modeled using a $193 \times 49 \times 81$ C-O topology mesh. This grid was found to be too coarse in the normal direction to adequately resolve the boundary layer. Both single-zone and two-zone grids were used to model the F/A-18 Forebody-LEX geometry.

The single-zone grid had 406,617 points, and the two-zone grid had 234,500 points. Laminar and turbulent (Baldwin-Lomax turbulence model) viscous flow fields were considered. Pressure-coefficient results for the $M = 0.24$, $\alpha = 30.3^\circ$, and $Re/ft = 885,000$ case compared well with flight test data. These cases required approximately 4 Cray-2 hours and 40 megawords of memory.

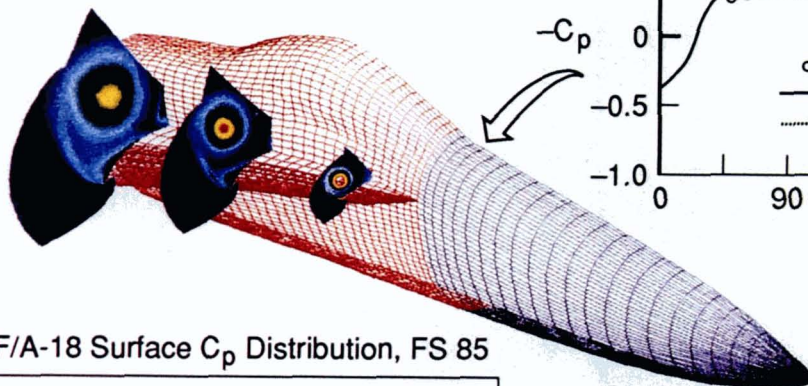
Significance

Future fighter aircraft will be required to maneuver within broader flight envelopes. The computational prediction of flows past complex geometries at high angles of attack will not only enhance the design of future aircraft but will aid in understanding the physics of such flows. The results obtained in this study are a significant step toward the Navier-Stokes analysis of full aircraft geometries at high angles of attack.

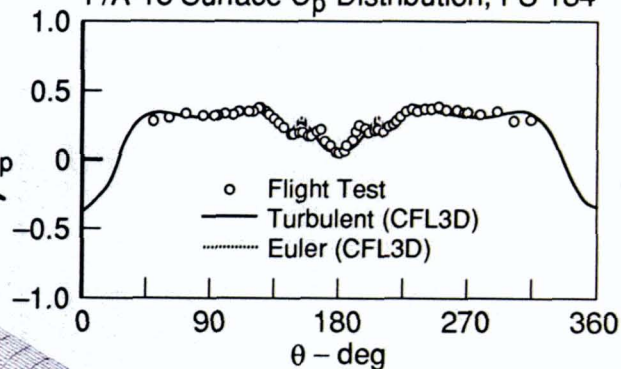
Future Plans

The AFVFM will be modeled with a grid that will adequately resolve the boundary layer. The F/A-18 geometry will be extended to include the wing. Comparisons with flight test data will continue.

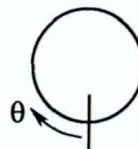
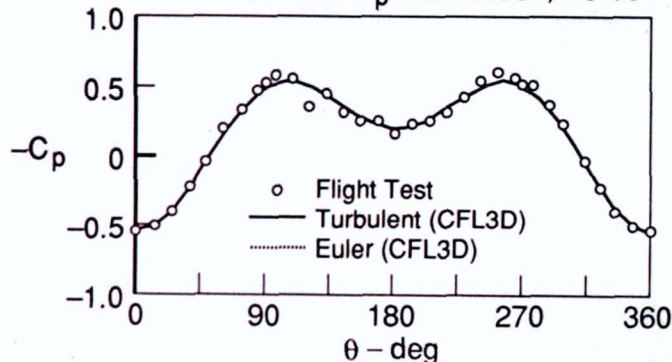
Zone 1: $35 \times 51 \times 50$
Zone 2: $35 \times 83 \times 50$



F/A-18 Surface C_p Distribution, FS 184



F/A-18 Surface C_p Distribution, FS 85



GP03-0007-216

Stagnation pressure contours for the F/A-18 Forebody-LEX geometry, two-zone grid; $M_\infty = 0.24$, $\alpha = 30.3^\circ$, $Re/ft = 885,000$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Direct Simulation of Compressible Turbulent Flows

Nagi N. Mansour, Principal Investigator
Co-investigators: J. Chen and G. Blaisdell
NASA Ames Research Center

Research Objective

To develop an understanding of the effects of compressibility on turbulent flows, and to develop a data base for compressible turbulent flows using direct simulation methods.

Approach

Three-dimensional time-dependent simulations of compressible turbulent flows are carried out using spectral methods. The effects of compressibility on the growth of wakes are studied by solving the compressible Navier-Stokes equations using Fourier methods in the streamwise and spanwise directions, and solving a mapped spectral in the cross-stream direction. The development of compressible homogeneous turbulent flows is studied by solving the Navier-Stokes equations using Fourier methods in all directions, in a moving grid.

Accomplishment Description

We found that the baroclinic and dilatational terms prevent the roll-up of vorticity at high Mach numbers, and thus reduce the growth rate of the wake. We found that vortex loops may or may not form, depending on the phase between the initial two-dimensional wave and the oblique wave perturbations. The turbulence energy in homogeneous flows under mean shear was found to decrease with increasing high Mach numbers.

We found that the streamwise component of the solenoidal part of the field grows as in the case of incompressible flows, while for the dilatational part of the field, the cross-stream component grows. This may be caused by the radiation of acoustic waves in the cross-stream direction.

Significance

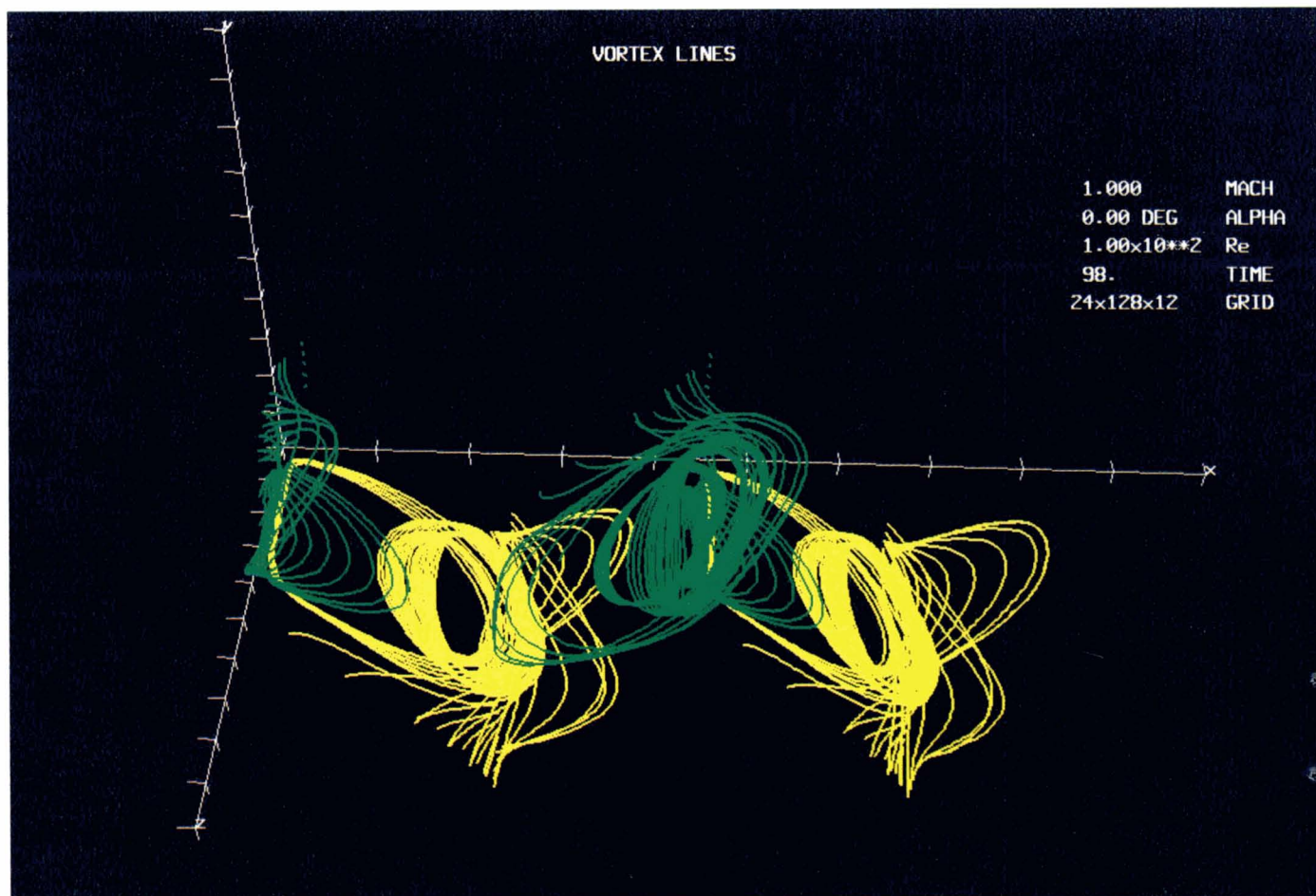
At high Mach numbers the effects of compressibility on the turbulence are felt through the dilatational part of the field and the baroclinic terms. The existing turbulence models that do not take the dilatational effects into account will fail for high Mach numbers. Data bases are available for testing modeling hypothesis and formulations.

Future Plans

We will continue to work on the development of wakes by studying the effects of random initial perturbations on the development of the large structures. Homogeneous flows under mean compression will be simulated using spectral methods.

Publications

Chen, J. H.; Cantwell, B. J.; and Mansour, N. N. "The Effect of Mach Number on the Stability of a Plane Supersonic Wake." *Physics of Fluids A*, June 1990.



Vortex lines in a plane wake, revealing the formation of vortex loops; $M = 1.000$, $\alpha = 0.00^\circ$, $Re = 1.00 \times 10^2$, time = 98.0, grid size $24 \times 128 \times 12$.

Space Shuttle Flow Field

Fred W. Martin, Jr., Principal Investigator

Co-investigators: Joseph Steger, Pieter Buning, Steve Labbe, Ray Gomez, Phil Stuart, Jeff Slotnick, Steve Parks, and Stan Johnson

NASA Johnson Space Flight Center/NASA Ames Research Center

Research Objective

To enhance the fidelity of the Space Shuttle ascent geometry by including additional components of the aft attach hardware between the orbiter and the external tank.

Approach

The F3D-Chimera scheme is used to obtain flow fields about the Shuttle launch vehicle in the Mach number range from 0.6 to 4.5, with particular emphasis in the transonic range.

Accomplishment Description

Two additional components of the orbiter-to-external-tank aft attach interface were added to the Shuttle launch vehicle grid system. The first item was a high-fidelity representation of the orbiter-to-external-tank vertical strut, and the second was an ellipsoid designed to approximate the frontal area of the 17-in.-diameter propellant feed lines. These features brought the modeled aft attach blockage to 85% of the blockage on the launch vehicle. An $M = 1.25$, $\alpha = -4.0$ solution was obtained with the new grid system. Integrated wing loads were slightly lower than, but compared favorably with, the Shuttle's IVBC-3 aeroloads data base for wing-root shear and bending. However, large discrepancies existed in the wing torsion comparison.

Significance

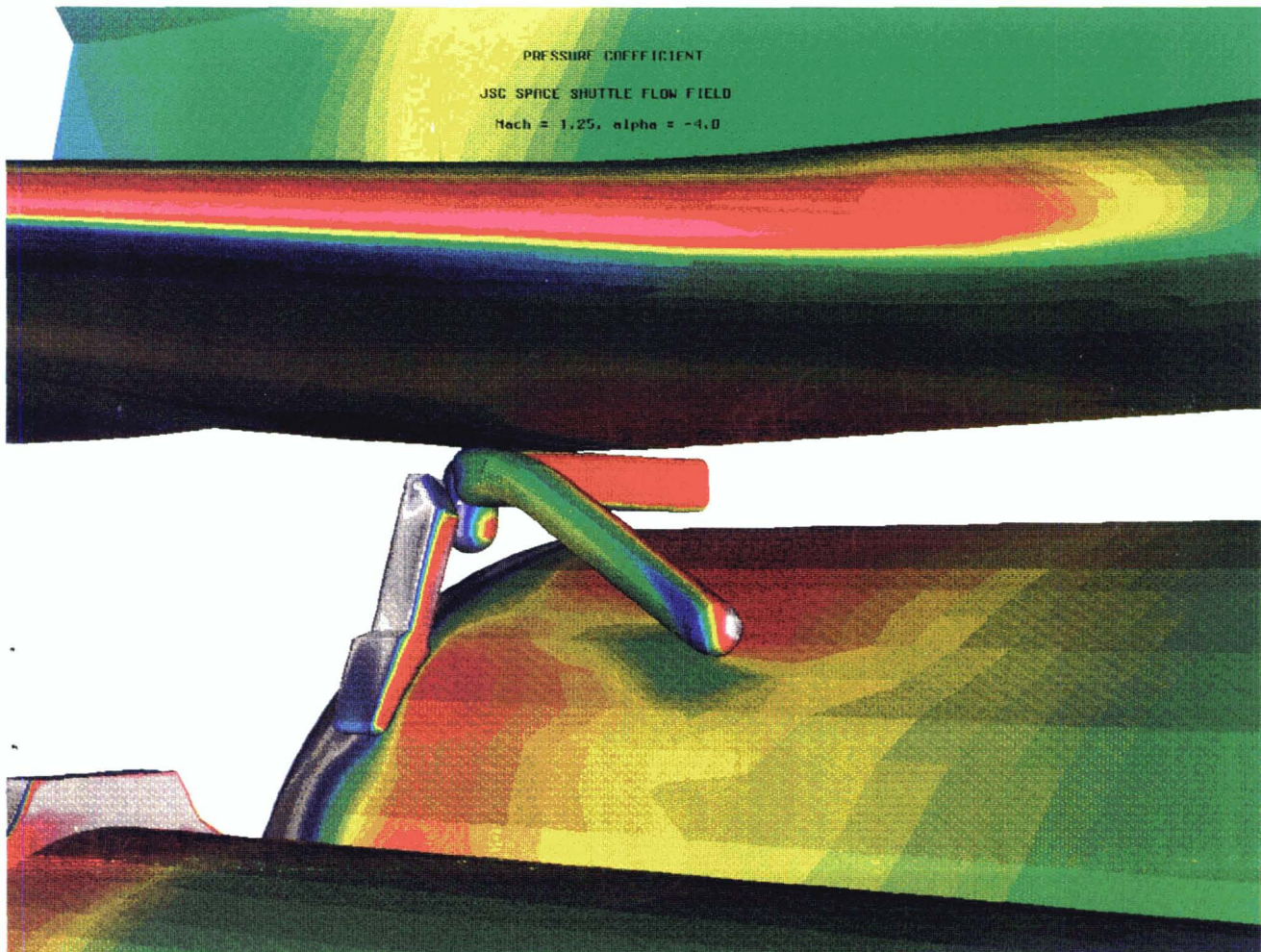
The Shuttle orbiter wing loads are adversely affected by the attach hardware between the Shuttle components. As a result, zero wing lift occurs on the negative- α side of the transonic-ascent flight envelope, instead of at the more benign center of the envelope. The correct modeling of the ascent vehicle's blockage and protuberances is instrumental to obtaining accurate simulations of the Shuttle ascent aerodynamic loads.

Future Plans

The fidelity of the aft attach hardware will be further improved by including the asymmetries caused by the liquid oxygen and liquid H₂ feed lines. Unfortunately, the corresponding calculations will not be able to take advantage of the plane-of-symmetry boundary condition, thus doubling the required computer resources.

Publications

Martin, F.; and Slotnick, J.; in collaboration with the Ames Space Shuttle Flow Field Simulation Group: Steger, J.; Buning, P.; Chiu, I.; Meakin, R.; Obayashi, S.; Rizk, Y.; and Yarrow, M. "Flow Computations for the Space Shuttle in Ascent Mode Using Thin Layer Navier-Stokes Equations." To be published in the AIAA Progress Series volume *Applied Computational Aerodynamics*.



Pressure coefficient for the Space Shuttle flow field; $M = 1.25$, $\alpha = -4.0^\circ$.

Solution of the Steady State Navier-Stokes Equations on Unstructured and Adaptive Meshes

Dimitri J. Mavriplis, Principal Investigator
NASA Langley Research Center/ICASE

Research Objective

To develop an accurate and efficient method for computing turbulent compressible flow about complex configurations.

Approach

The steady-state Navier-Stokes equations are solved on an unstructured triangular mesh with highly stretched elements in the boundary-layer and wake regions. Mesh adaptivity is used to enhance solution accuracy, and an unstructured multigrid technique is used to accelerate the convergence to steady state. An algebraic turbulence model specifically devised for use with multiple-body geometries and unstructured meshes is used.

Accomplishment Description

An accurate and efficient method for computing turbulent compressible flows has been devised. Accuracy is achieved by extensive use of adaptive meshing techniques. Efficiency is obtained by using a multigrid acceleration method and a relatively inexpensive algebraic turbulence model, both suitable for use on unstructured meshes. The multigrid algorithm accelerates the convergence to steady state by repeatedly time stepping on a sequence of coarse and fine unstructured meshes, each generated either globally or by adaptive refinement, and interpolating the flow variables back and forth between these meshes. The algebraic turbulence model operates on a set of local background structured meshes (one about each wall or wake boundary). At each time step, the flow variables are interpolated onto the background meshes, and the resulting eddy viscosities are interpolated back to the

global unstructured mesh. The interpolation patterns for both the multigrid and the turbulence model are determined in a preprocessing step, where an efficient tree-search algorithm is used. The turbulent compressible flow over a four-element airfoil configuration was computed using 62,000 grid points. A solution converged to engineering accuracy was obtained in 100 multigrid cycles, requiring approximately 40 minutes of Cray-2 CPU time and 16 megawords of memory. A total of 70 Cray-2 hours were used in the development and validation of this code.

Significance

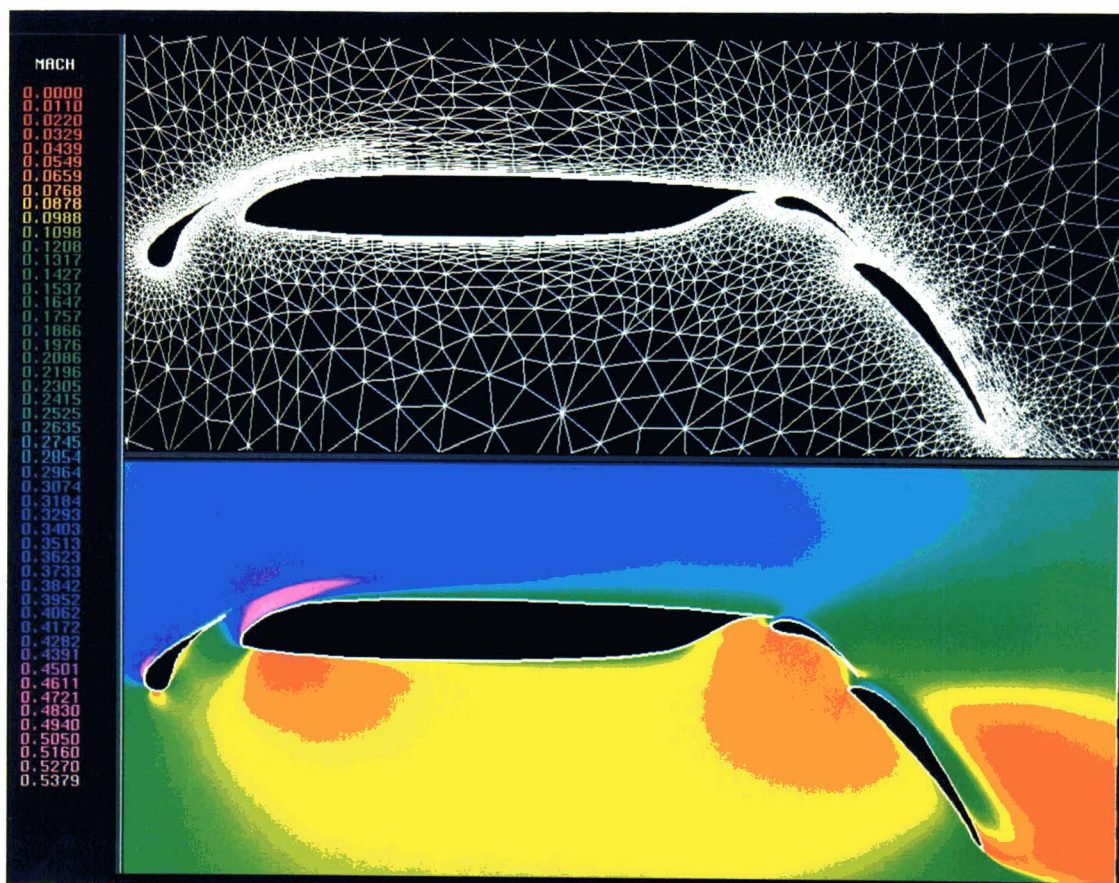
The ability to predict turbulent compressible flows about complex geometries such as high-lift multi-element airfoil configurations in two dimensions and full aircraft configurations in three dimensions is important to the aircraft industry.

Future Plans

Future work will study the effect of more sophisticated turbulence models on the solution accuracy and efficiency. The extension of these ideas to three dimensions is also under way.

Publications

1. Mavriplis, D. J., and Jameson, A. "Multigrid Solution of the Navier-Stokes Equations on Triangular Meshes." To be published in *AIAA J.*, 1990.
2. Mavriplis, D. J. "Euler and Navier-Stokes Computations for Two-Dimensional Geometries Using Unstructured Meshes." ICASE Rep No. 90-3, NASA CR-181977, Jan. 1990.



Solution of the steady state Navier-Stokes equations on unstructured and adaptive meshes.

Numerical Study of Planetary Atmospheres

Hans G. Mayr, Principal Investigator
Co-investigator: Kwing L. Chan
NASA Goddard Space Flight Center

Research Objective

The ultimate objective of this work is to understand the dynamics of the prevailing atmospheric circulations of the planets in the solar system. The questions of concern are: What mechanisms generate the alternating wind bands on Jupiter, Saturn, and Neptune? Is there a common origin? What generates the extremely large super-rotation of the atmosphere of Venus (60 times the rotation rate of the planet)? What are the differences and links among the atmospheres of the terrestrial planets and the outer planets? At first sight, these questions seem to address too broad a subject. In fact, some unifying principles that provide answers to these questions have been worked out, but mostly based on a quasi-linear theory (Mayr and Harris, *Astron. Astroph.* 121 (1983): 124; Mayr, Harris, and Chan, *Earth Moon and Planets* 30 (1984): 245; Mayr et al., *Adv. Space Sci.* 5 (1985): 63). Our numerical computations advance this theory into the nonlinear regime and check for the validity of some basic theoretical assumptions. The most important among them concerns the behavior of the Reynolds stresses in a rotating fluid. Numerical computation is the only viable way to study such problems.

Approach

Two- and three-dimensional codes are used to solve the Navier-Stokes equations for a compressible, stratified fluid. An alternating-direction implicit spectral code was developed to run on the Crays. This code describes the spherical geometry naturally and is extremely efficient for low-Mach number flows.

Accomplishment Description

An axisymmetric version of the spectral code was used to study the spin-up of the super-rotating Venusian atmosphere. This problem was chosen as a first step in our program because it is easier than the other problems and it can provide

us with valuable experience in adapting our codes to planetary applications. Since the problem is axisymmetric, the demand on memory is so far not high (about 10 megawords). However, the CPU requirement is already substantial. The difficulty lies in the extremely long physical relaxation time. The Venusian atmosphere needs a few hundred years to spin up. Even running at a CFL number of 103, the code needs about one million steps to converge; that takes about 2 hours on the Cray Y-MP.

Significance

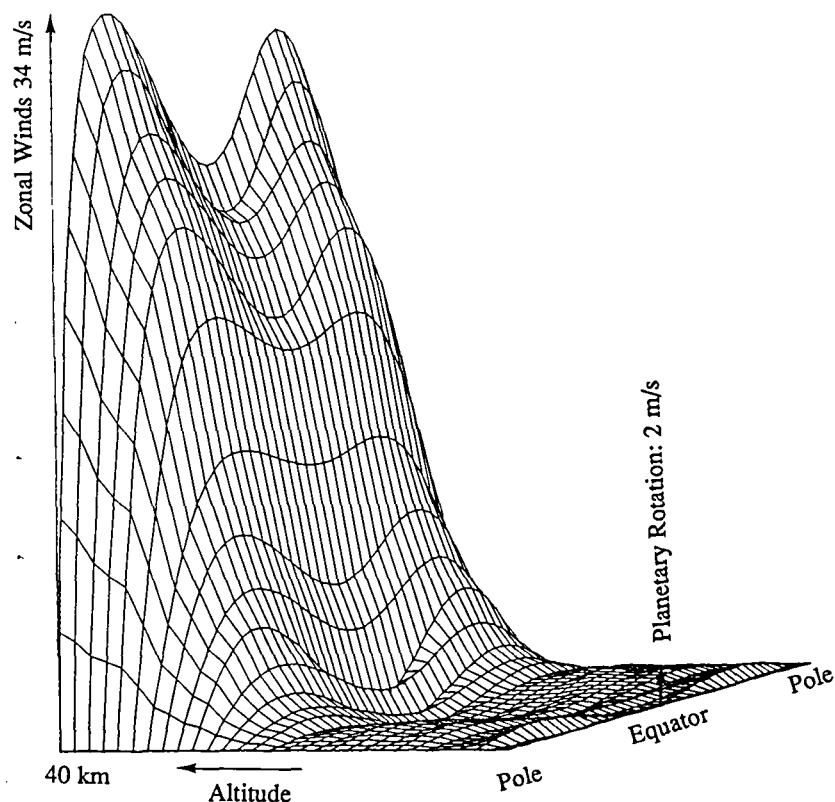
The resulting zonal wind from the numerical model is within a factor of two of that observed on Venus. Since there are still many uncertainties about the atmospheric conditions of Venus, and some parameters can be adjusted, this calculation demonstrates that the super-rotation of the Venusian atmosphere can in principle be explained. The parametric behavior of the model is in line with the expectation from the quasi-linear theory. Numerically, it shows that more realistic three-dimensional simulations are feasible.

Future Plans

We plan to carry out three-dimensional calculations to study the behavior of the Reynolds stresses (especially the horizontal ones) in order to check whether they are diffusive as assumed in the axisymmetric model and whether they are large enough to transport sufficient angular momentum from high latitudes to the equatorial region, where large zonal winds are observed.

Publications

Chan, K. L., and Mayr, H. G. "Modeling the Circulation in a Venus-like Atmosphere." Presented at the 1990 Spring AGU Meeting, Baltimore, MD, May 1990.



Venus-like atmospheric super-rotation.

National Aero-Space Plane Propulsion Flow-Path Analysis and Code Certification

Charles R. McClinton, Principal Investigator

Co-investigators: R. Bittner, G. Bobskill, R. Hawkins, T. Jentink, P. Kamath, M. Mao, G. Mekkes, and D. Riggins

NASA Langley Research Center

Research Objective

To certify computer codes for National Aero-Space Plane (NASP) scramjet flow-field evaluation and test support.

Approach

The SPARK3D, CFL3D, and GASP codes are applied to selected "unit" and "component" experimental problems identified as typical of NASP inlet, combustor, and nozzle flows. Calculated performance parameters and flow fields are compared with experimental results to obtain confidence in the ability of computational fluid dynamics to model large-scale flight conditions.

Accomplishment Description

Numerical analysis of three simulated scramjet nozzle experiments, nine transverse fuel injectors with no reaction, one direct connect combustor experiment (illustrated in the figure) and two inlet experiments provided an enhanced understanding of engine flow fields, a direct comparison with experimental measurements, an understanding of solution grid density and computer memory and CPU requirements, and an estimate of numerically generated engine thrust performance accuracy. The photograph illustrates the hydrogen mass fraction distribution in a direct connect combustor, just downstream of a swept-ramp fuel injector. Calculated wall pressures over the entire 1.3-m length of the direct connect experiment are in reasonable agreement with experimental measurements, and the calculated combustion efficiency is

within 5 to 10% of the experimentally determined value. Other performance prediction accuracies are discussed in the publications.

Significance

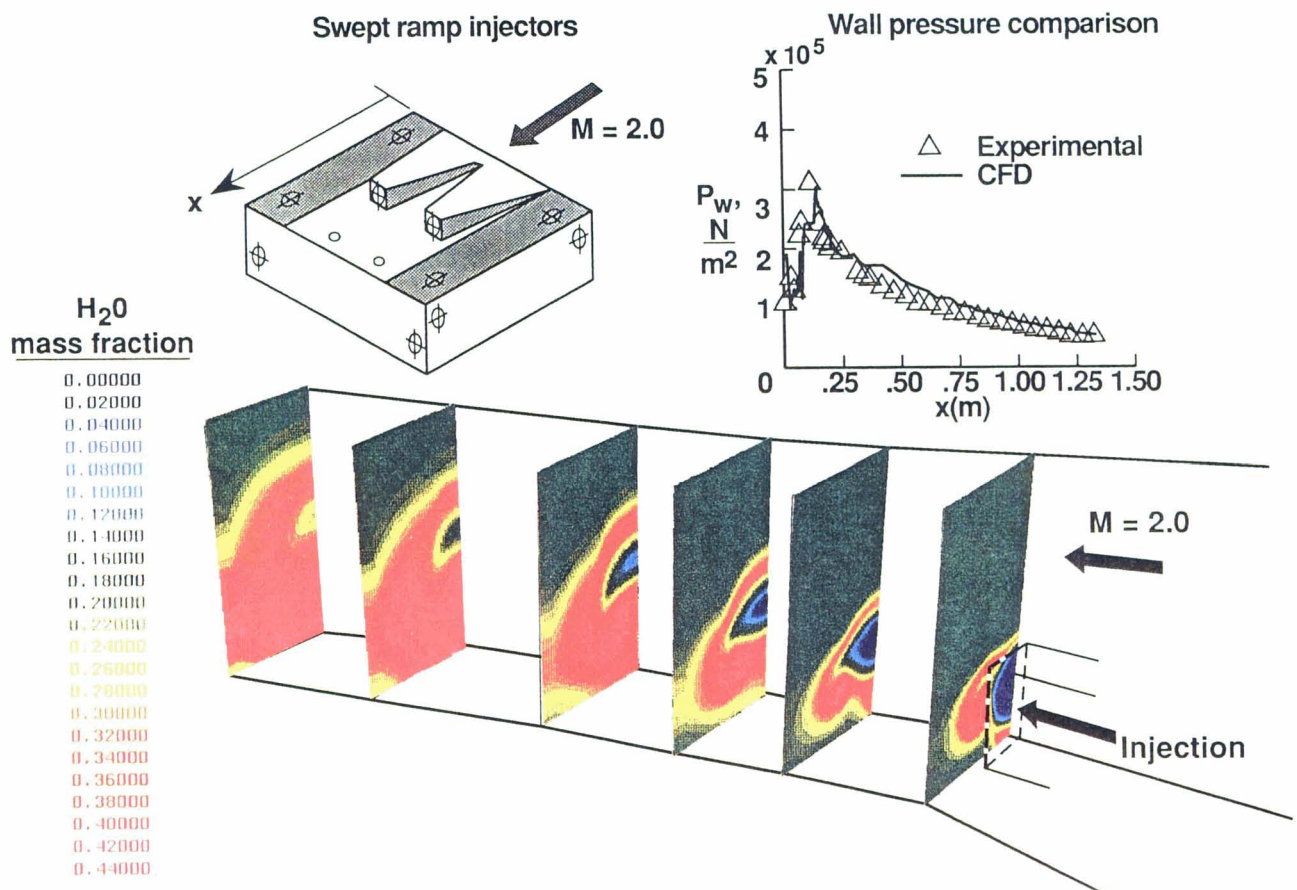
The capability developed under this project has provided an understanding of the capabilities and limitations of numerical analysis of large-scale scramjet flow fields and has clearly demonstrated the value of this numerical analysis in providing design support for hypersonic air-breathing vehicles.

Future Plans

We will continue more complex and higher speed inlet, combustor, nozzle, and complete airframe integrated engine-vehicle analysis using and supporting ground and flight tests.

Publications

1. Drummond et al. Paper 17, 6th NASP Symposium, Apr. 1989.
2. Riggins et al. Paper 25, 6th NASP Symposium, Apr. 1989.
3. Mekkes et al. Paper 55, 6th NASP Symposium, Apr. 1989.
4. Drummond et al. AIAA Paper 89-2794, 1989.
5. Richardson et al. SAE Paper 89-2313, 1989.
6. Riggins et al. NASP CR-1043, Apr. 1989.
7. Riggins et al. Paper 56, 7th NASP Symposium, Oct. 1989.
8. Riggins et al. AIAA Paper 90-0203, 1990.



Mixing reaction with enhancement.

Aerodynamic Flows about High-Performance Rotor Blade Tips

W. J. McCroskey, Principal Investigator

Co-investigators: J. D. Baeder, G. R. Srinivasan, and E. P. N. Duque

U. S. Army Aeroflightdynamics Directorate—AVSCOM/NASA Ames Research Center

Research Objective

The objective of this research is to develop and validate accurate, user-oriented, viscous computational fluid dynamics (CFD) codes for three-dimensional, unsteady aerodynamic flows about advanced-geometry rotor blade tips. These codes are then used to study the fluid physics of tip vortex formation and roll-up from complex tips, and to develop improved tip shapes for high-performance rotorcraft.

Accomplishment Description

Thin-layer Navier-Stokes solutions were used to further analyze the nonrotating aerodynamic characteristics of the British Experimental Rotor Program (BERP) blade. In all flow regimes explored, the computations predicted that the blade had many favorable aerodynamic characteristics in comparison with conventional helicopter rotor blades. The computed results exhibited a good qualitative comparison with the wind tunnel tests performed by Westland Helicopters Ltd., as shown in the accompanying figure. The solutions required approximately 25 megawords of memory and 25 hours of Cray-2 CPU time per solution.

Significance

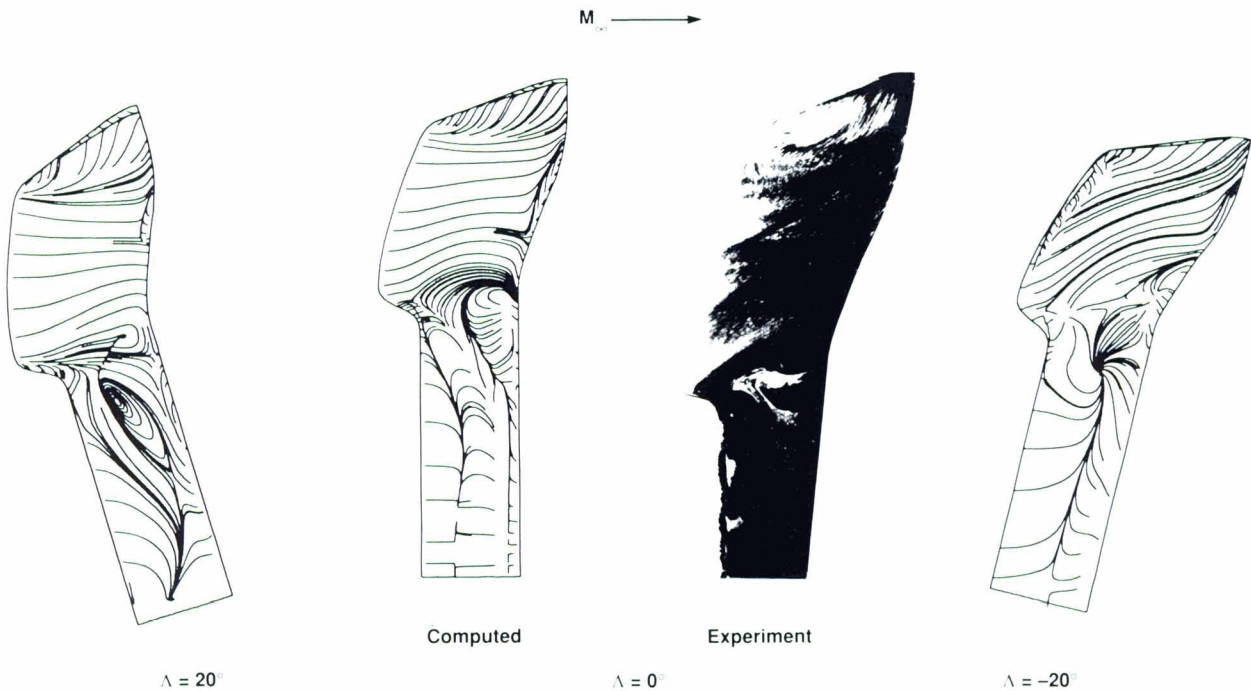
This work provides a source of aerodynamic data for the BERP blade. It demonstrates the ability to predict the viscous three-dimensional effects of complex helicopter blade tips. It also illustrates the ability of CFD to provide a better understanding of complex aerodynamic geometries, not easily attained by experiment, and it provides a new tool the rotorcraft community can use to analyze and design new blade tips.

Future Plans

The group will continue to develop more accurate and efficient flow simulation codes. Future computations will include computations of other advanced helicopter blade tips in both hover and forward flight. These studies will allow new tips to be designed that will improve helicopter performance.

Publications

1. Duque, E. P. N. "A Numerical Analysis of the British Experimental Rotor Program Blade." NASA TM-102247, Nov. 1989.
2. Brocklehurst, A., and Duque, E. P. N. "Experimental and Numerical Study of the British Experimental Rotor Program Blade." AIAA Paper 90-3008, Aug. 1990.



Comparison of computed results with wind tunnel tests; $\alpha = 20^\circ$, $M = 0.20$, $Re = 1.5 \times 10^6$.

Airloads and Acoustics of Rotorcraft

W. J. McCroskey, Principal Investigator

Co-investigators: J. D. Baeder, E. P. Duque, and G. R. Srinivasan

U. S. Army Aeroflightdynamics Directorate—AVSCOM/NASA Ames Research Center

Research Objective

To develop and validate accurate, user-oriented, viscous computational fluid dynamics (CFD) codes (with inviscid options) for three-dimensional, unsteady aerodynamic flows about arbitrary rotorcraft configurations; developing the capability to calculate the associated acoustic field is included.

Approach

Unsteady Euler and thin-layer Navier-Stokes codes are being developed, adapted, and extended to simultaneously capture the aerodynamics and the acoustics of rotorcraft. Accuracy and stability are achieved by utilizing Roe upwind-biasing with third-order-accurate in space MUSCL-type limiting. Efficiency is obtained with an LU-SGS implicit operator on the left-hand side, combined with Newton iterations. In order to properly capture the acoustics, the grid is extended to a distance several rotor radii away from the center of rotation.

Accomplishment Description

The previous investigation of the aerodynamics of isolated rotor blades in hover was extended to capture the high-speed impulsive (HSI) noise of a nonlifting rotor. The accompanying figure shows the pressure contours and grid in the plane of the rotor for a tip Mach number of 0.92. The time history of the pressure disturbance at a location 2.18 rotor radii from the center of rotation is shown in the inset; the agreement with experimental data is excellent. Fine-grid Euler computations, using over 400,000 grid points, required about 20 megawords of central memory and 5 Cray-2 hours of CPU time per solution. Similar agreement with experiment was shown for a range of Mach numbers, out to a distance of 3 rotor radii (over 42 chord lengths). A second code was developed to capture the vortical structure of the rotor wake without ad hoc modeling, thus facilitating the viscous simulation of multiple lifting

blades in hover. Preliminary coarse-grid Navier-Stokes calculations, using about 100,000 grid points, required about 10 megawords of central memory and 3 Cray-2 hours of CPU time per solution. The computed near-wake effects of the vortical wake show good agreement with experimental data.

Significance

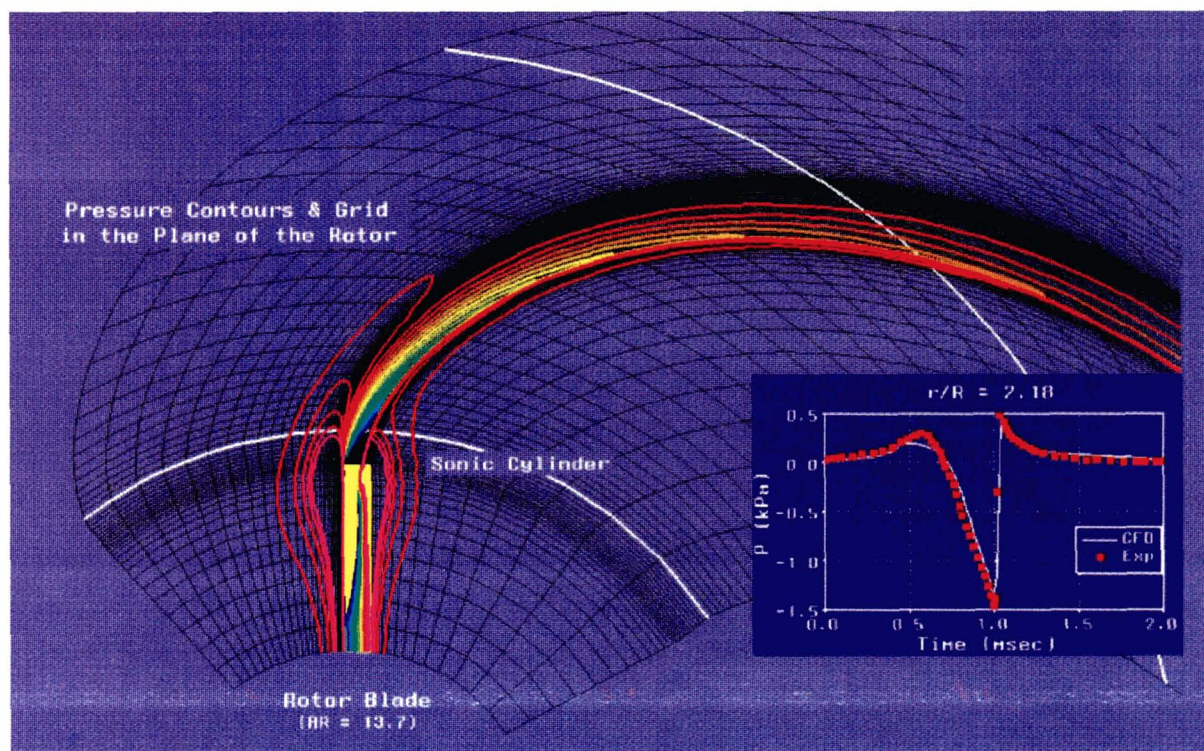
The ability to simulate accurately the aerodynamics and acoustics of rotorcraft will enable significantly better and quieter vehicles to be designed at lower cost, with less expensive testing and less risk. This is the first time that a purely CFD method has been used to calculate HSI noise; it has resulted in agreement with experiment that is vastly improved compared with previous analytical methods, which are usually off by almost a factor of two in predicting the peak negative pressures.

Future Plans

The research will continue in order to develop more accurate and efficient simulations of rotor aerodynamics and acoustics, with particular emphasis given to combining the above capabilities to investigate the rotor wake and its influence on blade airloads and acoustics for various blade planform and airfoil shapes. This will necessitate the development of multizones and/or adaptive grids, especially for forward flight.

Publications

1. "The Computation and Analysis of Acoustic Waves in Transonic Airfoil-Vortex Interactions." Ph.D. Thesis, Stanford Univ., Sept. 1989.
2. "Computational Analysis of Transonic Airfoil-Vortex Interaction Acoustics and Rotorcraft CFD." NASA 7th Annual Review of Helicopter Noise Reduction Program, Oct. 1989.
3. "Unsteady Interaction of a Rotor with a Vortex." AIAA Paper 89-1848, June 1989.



Noise resulting from thickness for a nonlifting hovering rotor, Euler solution; $M_{tip} = 0.92$.

Tilt-Rotor Aerodynamic Interactions

W. J. McCroskey, Principal Investigator
Co-investigators: V. Raghavan and S. Stanaway
NASA Ames Research Center

Research Objective

To apply Reynolds-averaged Navier-Stokes codes to interactional aerodynamic problems associated with high-speed rotorcraft, including tilt-rotor aircraft.

Accomplishment Description

Proposed designs for a high-speed rotorcraft involve some hybrid of a helicopter and an airplane—for example, a tilt rotor. One unique aspect of this application is the presence of a rotating grid around the rotor blades combined with a fixed grid about a wing and fuselage. The approach taken was to validate the Navier-Stokes codes for individual vehicle elements (a fuselage and/or a wing at 90° angle of attack), place these elements in the presence of a modeled rotor on a fixed grid, and compute the flow for the combined rotor and body. The full-scale interactional aerodynamic test, whose set-up is shown in the accompanying figure, was chosen as a test case for code validation. The results of a Reynolds-averaged Navier-Stokes calculation is also shown; the color mapping depicts the pressure distribution on the body, and the lines indicate the surface skin friction vectors on the isolated body. Another component studied extensively was the tilt-rotor wing airfoil at an angle of attack of -90° , showing high-frequency shedding for laminar and turbulent flows. The rotor was modeled initially by an actuator disk, which was treated implicitly in the Navier-Stokes environment as an internal boundary condition in a cylindrical grid, using the fortified Navier-Stokes technique. Preliminary calculations for the fuselage body used 210,000 points, requiring about 4 hours of Cray-2 time, and typical bluff-body calculations used 20,000 points, requiring 30 minutes of Cray-2 time per solution.

Significance

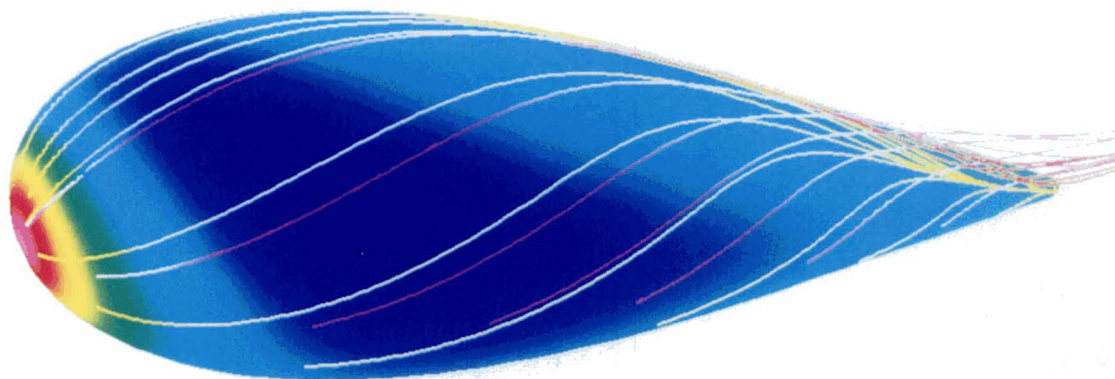
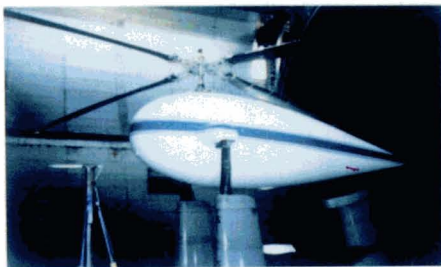
Three-dimensional fuselage calculations have shown skin friction patterns similar to experimental results. The two-dimensional bluff-body predictions showed some success and also pointed to the areas that need further study, namely, turbulence models for unsteady shear flows. The actuator disk was implemented successfully in the Navier-Stokes environment. This work demonstrates the usefulness of Reynolds-averaged Navier-Stokes equations for complex viscous environments, and lays the groundwork for interactive aerodynamic calculations.

Future Plans

Future plans include more extensive code validation for the isolated body, followed by rotor/body interaction calculations for a fuselage and actuator disk. Similarly, we will compute the three-dimensional wing at 90° and combine it with an actuator disk to predict the tilt-rotor download in the take-off configuration. Although the actuator disk model assumes a uniform disk loading, this is not necessary in our approach, and the next step will be to include a nonuniform disk-loading model. The potentially complex grid generation task typically involved for a complex geometry such as the rotor and body combination will be simplified by using the overset grid technique, Chimera. Other work in progress includes better turbulence modeling of massively separated flow, and code performance improvements.

Publications

Raghavan, V.; McCroskey, W. J.; Van Dalsem, W. R.; and Baeder, J. D. "Calculations of the Flow Past Bluff Bodies, Including Tilt-Rotor Wing Sections at $\alpha = -90$ degrees." AIAA Paper 90-0032, Jan. 1990.



Turbulent flow; $M = 0.18$, $\alpha = 17^\circ$, $Re = 27.0 \times 10^6$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Incipient Leading-Edge Separation

S. Naomi McMillin, Principal Investigator
NASA Langley Research Center

Research Objective

To computationally determine the effect of leading-edge radius and camber on the formation of incipient leading-edge separation over the lee side of delta wings at supersonic speeds.

Approach

A computational parametric study is conducted on a 65°-swept, conical delta wing at $M = 1.6$, using CFL3D—a Navier-Stokes computational code.

Accomplishment Description

The conical geometries examined in this study were three uncambered 65°-swept delta wings that varied in leading-edge radius only and three 65°-swept rounded-leading-edge delta wings that varied in spanwise camber. In the previous year, conical Navier-Stokes solutions were obtained using the turbulent boundary-layer model at $M = 1.6$ and $Re = 1.0 \times 10^6$, 2×10^6 , and 5×10^6 . The results showed that applying leading-edge radius and/or camber on a delta wing can delay the onset of leading-edge separation such that leading-edge separation occurs at a higher angle of attack. These computations have been repeated using the laminar-boundary-layer model in order to study the effect of boundary-layer model on the incipient separation characteristics of these geometries. A typical solution needed approximately 0.6 hours of Cray-2 time and 4 megawords of memory. Typical results are presented in

the figures in the form of crossflow Mach number contours. The solution in the left-hand figure is the turbulent-boundary-layer solution for the 10° camber wing at 8° angle of attack. The code predicted attached flow with an isentropic compression occurring inboard of the leading edge. The laminar-boundary-layer solution presented on the right predicts a leading-edge separation bubble. This comparison between laminar- and turbulent-boundary-layer solutions suggests that the effect of leading-edge radius and/or camber in delaying the onset of leading-edge separation is less effective with a laminar-boundary-layer condition on the surface of the wing.

Significance

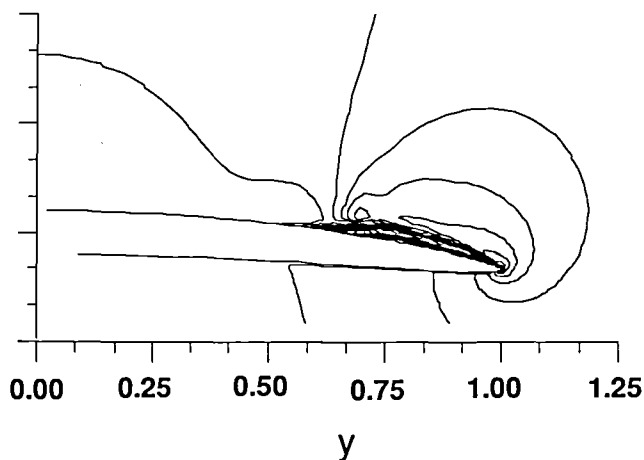
The code CFL3D has been used to quantify the effects of leading-edge radius, camber, Reynolds number, and possibly boundary-layer condition on incipient leading-edge separation over delta wings.

Future Plans

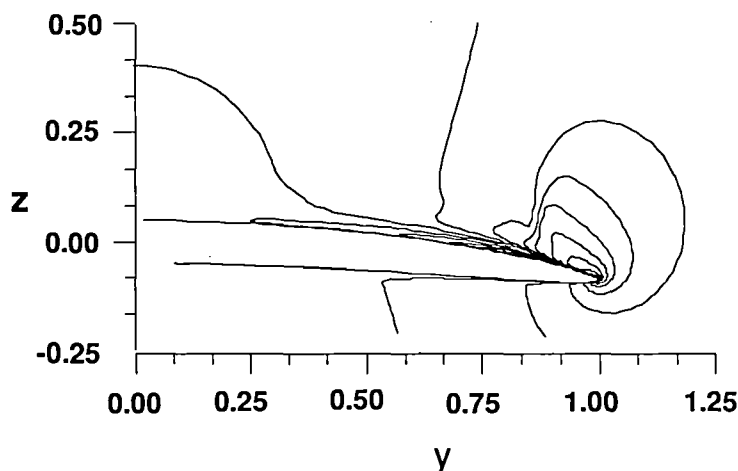
Wind tunnel models will be tested and experimental data will be compared with computational results. Then the computational study will be extended to further investigate the effect of boundary-layer model on incipient separation.

Publications

"A Computational Study of Incipient Leading-Edge Separation at $M = 1.60$." Presented at the AIAA Applied Aerodynamics Conference, Portland, OR, Aug. 1990.



Turbulent boundary layer.



Laminar boundary layer.

Crossflow Mach number contours, showing the effect of the boundary-layer model on computational results; $M = 1.6$, $\alpha = 8^\circ$, $Re = 1 \times 10^6$.

Multiblock, Multigrid Method for the Solution of the Three-Dimensional Euler Equations

N. Duane Melson, Principal Investigator

Co-investigators: Frank E. Cannizzaro, Alaa Elmiligui, and E. von Lavante
NASA Langley Research Center

Research Objective

To develop a computer program that would aid in the study of rectangular nozzle flows exhausting into different free-stream conditions.

Approach

An explicit upwind method was used to solve the finite-volume formulation of the Euler equations. Multigrid acceleration was implemented to increase the rate of convergence. The computer program was transformed into a multiblock structure to provide flexibility for analyzing different geometric configurations.

Accomplishment Description

Two types of upwind methods were investigated to solve the body-fitted finite-volume formulation of the Euler equations: van Leer's flux vector splitting and Roe's flux difference splitting. Both used MUSCL-type differencing. A modified Runge-Kutta time-stepping scheme was used to advance the solution. A multiblock capability was used to compute the nozzle flow shown in the accompanying figure. The flow conditions were $M_\infty = 3.0$, $M_{jet} = 1.99$, $P_{jet}/P_\infty = 8.96$, and $T_{jet}/T_\infty = 3.55$.

Significance

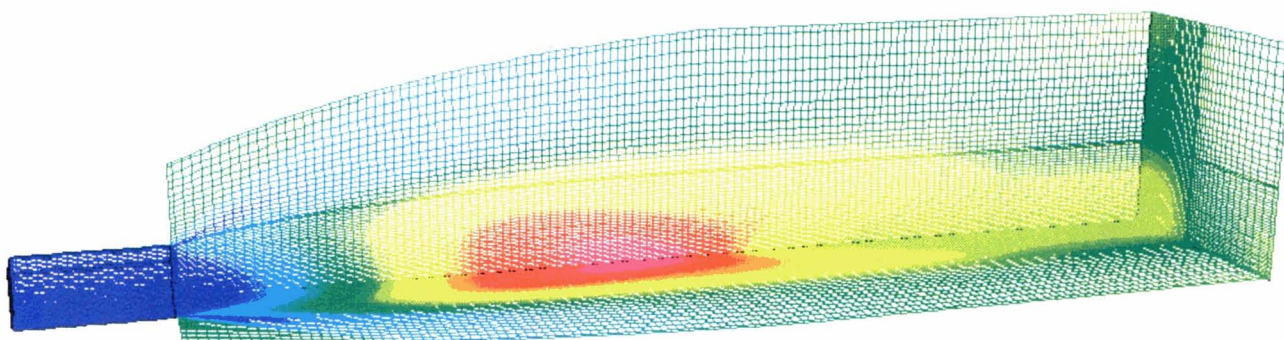
The multiblock structure provides a computer program that is independent of grid topology. Thus, the same computer program can be used to solve internal flows (such as nozzle flows, which may be an H-H grid topology) and external flows (such as flow over a wing, which may be a C-H, a C-O, or an O-O grid topology) without requiring changes to the program. Inputs to the computer program require that boundary and interface conditions of each face of each block be specified, therefore allowing any type of grid topology to be handled.

Future Plans

We plan to increase the range of nozzle flow conditions that can be solved, improve the computational efficiency of the computer program by investigating various types of residual smoothing, and include viscous terms in the governing equations.

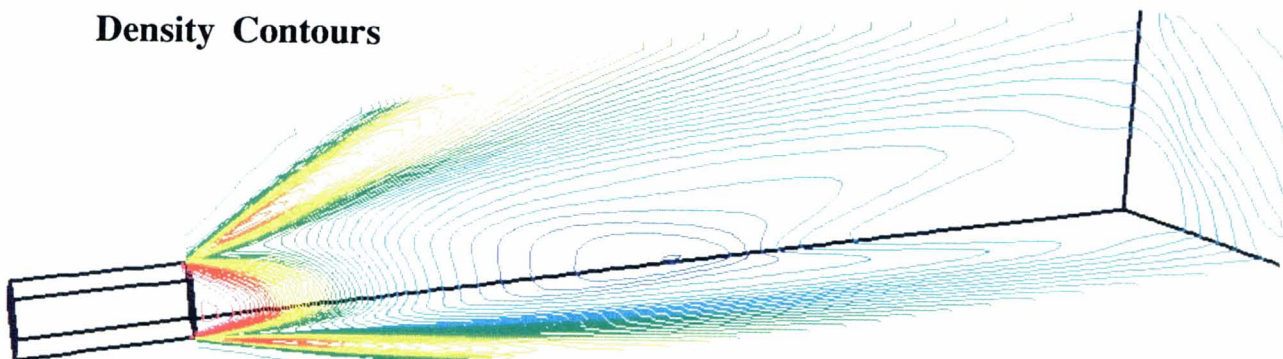
Publications

Cannizzaro, Frank E.; Elmiligui, Alaa; Melson, N. Duane; and von Lavante, E. "A Multiblock Multigrid Method for the Solution of the Three-Dimensional Euler Equations." AIAA Paper 90-0105, Jan. 1990.



Mach Contours On Computational Grid

Density Contours



(Top) Nozzle-flow Mach contours on a computational grid. (Bottom) Nozzle-flow density contours.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Computational Fluid Dynamics Analysis of Advanced Turboprop Configurations

John E. Melton, Principal Investigator
Co-investigators: Ronald G. Langhi and Brian A. Nishida
NASA Ames Research Center

Research Objective

To develop advanced computational fluid dynamics tools for the design and analysis of transport aircraft configurations that use advanced turboprop or propfan propulsion systems.

Approach

Advanced grid-generation techniques and a modified Euler flow solver are used to model complex wing/nacelle geometries and to predict accurately the effects of the propeller slipstream on the aerodynamic performance of the configuration.

Accomplishment Description

A three-dimensional elliptic grid-generation program was used on the Cray-2, Navier, to produce volume grids up to 1.2 million points in size about various fuselage/wing/nacelle transport aircraft configurations where the point density in regions of interest and the orthogonality of grid lines intersecting the geometry surface were especially critical. Typical CPU time and memory requirements were on the order of 2400 seconds and 15 megawords, respectively. To study the effects of the propeller slipstream, an actuator disk algorithm was incorporated into Jameson's multistage scheme for the Euler equations. Calculated velocities over a propeller/nacelle test rig were computed and found to be in good agreement with laser Doppler velocimeter data from a NASA Lewis

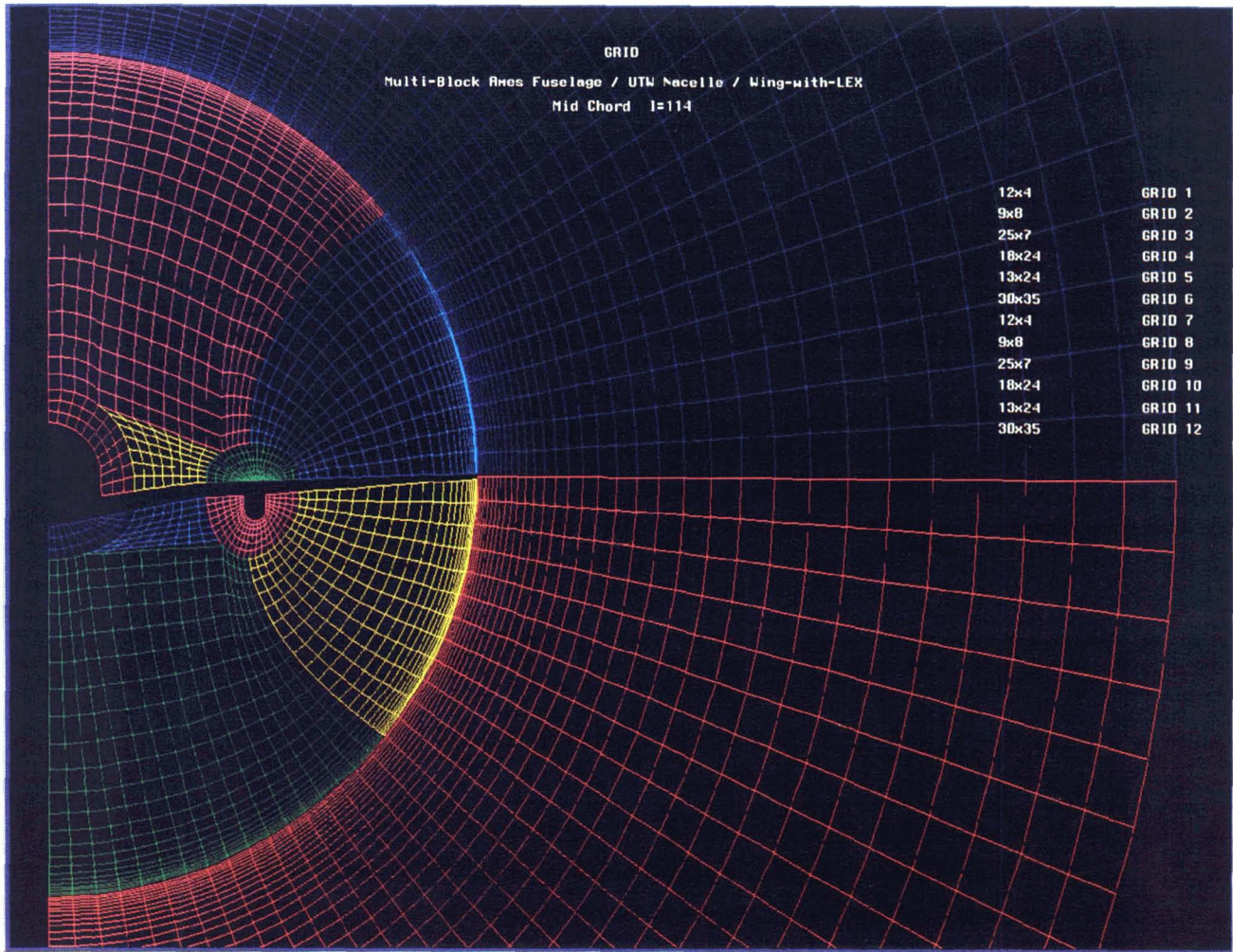
experiment. More detailed computations over a wing/body/nacelle/propeller were done, and the results compared well with NASA Ames 11-Foot Wind Tunnel data. Further improvements in the Euler code included multiple-block capabilities that enabled the program to handle general, complex geometries; exploit the fast input/output of Reynold's SSD; and take advantage of the multiple-block grid-generation packages now available. An unstructured, hexahedral-based Euler code that used grid enrichment was also developed and tested on a variety of complex configurations.

Significance

Future advanced turboprop aircraft will require a high degree of aerodynamic integration in order to fully exploit the fuel efficiency of the propfan propulsion systems. Computational analysis can provide additional understanding of important flow-field phenomena and insights that are difficult to obtain experimentally about design considerations.

Future Plans

More testing and verification of the Euler codes is currently under way so that they can be routinely used as design tools. The grid-generation package is mature and is being used on a variety of other programs.



Grid for a multiblock Ames fuselage/UTW-nacelle/wing-with-LEX configuration; midchord I = 114.

Vortex-Induced Nonlinearities on Submarines

Michael R. Mendenhall, Principal Investigator

Co-investigator: Stanley C. Perkins, Jr.

Nielsen Engineering & Research, Inc.

Research Objective

The objective of this work is to investigate an analytical procedure to predict the nonlinear fluid mechanics characteristics of modern attack submarines in steady and unsteady maneuvers involving high incidence angles and high angular rates. Effects of flow separation and shed vorticity are to be included, and the resulting method is to be applicable to generic submarine configurations for a wide range of flow conditions and Reynolds numbers.

Approach

A hydrodynamic prediction method is coupled with a six-degree-of-freedom (6-DOF) equation-of-motion solver to calculate the motion of a submarine configuration. The calculation requires only the submarine geometry and mass characteristics, initial flow conditions, and time histories of propeller rotation rate and fin deflections. Components of the submarine are modeled with potential flow models; specifically, the axisymmetric hull is represented by axis singularities and the lifting surfaces are represented by a panel method. Flow separation from the hull is calculated using modified Stratford criteria, and hull shed vortices are modeled by clouds of discrete vortices. Trailing vortices from the lifting surfaces are treated as free vortices aft of the trailing edge of the surface. Starting at an initial time with a given set of flow conditions, a marching procedure is used to calculate pressures, flow separation, and forces and moments on the submarine components. The resulting total submarine forces and moments are used in conjunction with a 6-DOF equation-of-motion solver to calculate the motion of the submarine over a small time interval, in order to produce a new orientation of the vehicle and new flow conditions. The vortex wake is allowed to move under the influence of the changing flow conditions during the time interval, and the modified vortex field influences the pressure distribution on the vehicle and thus affects subsequent separation characteristics. New vortices are added to the flow field, new forces and moments are calculated, and the submarine motion is predicted for another time interval. The procedure is continued for a specified time or maneuver.

Accomplishment Description

The above code and prediction methods were applied to a modern attack submarine configuration under various maneu-

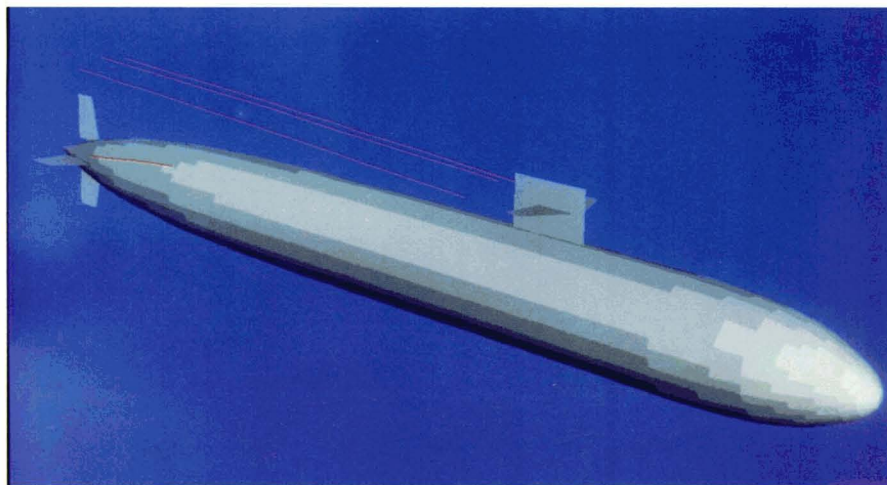
vering conditions, and the predicted results were compared with available sea trial results. The calculation procedure worked well and a number of detailed results were obtained. For example, the vortex lag caused by the delay between a vortex being shed from the hull and its later influence on tail control surfaces proved to be a critical component of the flow phenomena that dictate the behavior of the maneuvering submarine. For one maneuver condition, the time history of the forces on the four tail fins showed a vortex-induced imbalance that created a rolling moment on the submarine and influenced the calculated motion. A typical maneuver lasting approximately 25 seconds required between 0.5 and 1.0 Cray Y-MP hours and 2 megawords of memory. The accompanying figure shows traces of vortex centroids from the sail, sailplanes, and deck of a generic submarine configuration in steady flow conditions. The keel side vorticity cannot be seen in this view.

Significance

Modern attack submarines in steady and unsteady maneuvers involving high incidence angles and high angular rates have fluid mechanic characteristics dominated by flow separation, vorticity, and unsteady lag effects. The design and analysis of advanced submarines requires computational methods that can represent these complex flow phenomena, to produce the highly nonlinear forces and moments acting on maneuvering submarines. An improved analytical capability will provide a means to investigate stability and control effects, particularly in regions of vortex-dominated flows. One use of this capability is to provide insight into the complex problem of vortex interference on control surfaces, especially in a time-dependent flow field.

Future Plans

The objective of the second phase of this computational effort is to study further the direct calculation simulation approach for submarines and to continue to apply the method to the prediction of submarine trajectories in unsteady maneuvers. A number of computational and physical problems became evident as a result of the first phase of this study. For example, it is now apparent that the unsteady character of the separation line on the hull may be different from the known separation line in steady flow. This is one important area requiring further study.



Traces of vortex centroids from the sail, sailplanes, and deck of a generic submarine configuration in steady flow.

Standing Oblique Detonation Waves

Gene P. Menees, Principal Investigator

Co-investigators: Henry G. Adelman and Jean-Luc Cambier
NASA Ames Research Center

Research Objective

The ultimate objective of this work was to demonstrate the proof-of-concept of an oblique detonation wave engine (ODWE) for hypersonic air-breathing propulsion. The work included an experimental study performed in an arc-heated wind tunnel, and a series of analytical studies. The development and validation of a numerical capability for the time-accurate simulation of detonations and supersonic combustion was an essential part of the program.

Approach

Two- and three-dimensional Navier-Stokes codes were developed that used a second-order, upwind (total-variation-diminishing) scheme. The code used multiple-species convection coupled with detailed chemical kinetics in a time-accurate fashion, along with an explicit, operator-splitting approach. Fickian and laminar diffusion processes were also incorporated. The codes used a fully conservative grid-patching algorithm to allow the modeling of complex geometries and to increase the computational efficiency for steady-state as well as unsteady calculations.

Accomplishment Description

The code was used to successfully simulate oblique detonation waves attached to a ramp, and the results were in accord with analytical expectations. It was also used to model the standing detonation waves in various mixing conditions and to study the wave patterns expected in the experiment. It was seen that in the limit of low mixing, the detonation reduces to shock-induced combustion, with a flame anchored at the primary shock. It was also seen that the combustion can strongly amplify initial perturbations in the shock curvature, thereby possibly leading to a bootstrap effect of enhanced mixing and combustion. The code was also used to model the fuel injection from struts, according to the experimental design, in order to estimate the positions of shock impingements and fuel penetration and mixing. Finally, the code was validated by comparing the results with previous experimental studies of shock/flame interactions done in a ballistic range. The exact reproduction of the position of flame/shock decoupling, as indicated in the accompanying figures, is a critical test of the accuracy of the coupling between chemistry and convection algorithms in the code. This calculation was done on

36,000 grid points, with 9 chemical species and 44 elementary (one-step) chemical reactions, and required 12 hours of Cray-2 time and less than 10 megawords of memory.

Significance

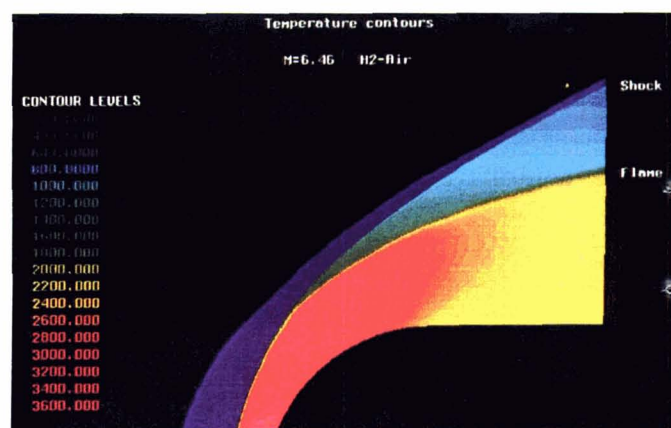
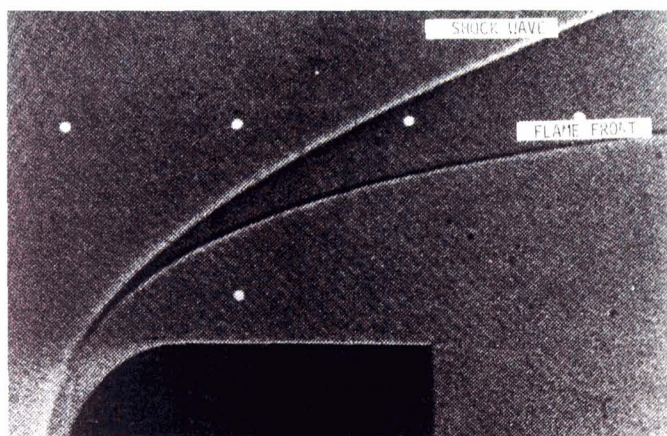
The ODWE represents an alternative concept to the scramjet for hypersonic air-breathing propulsion. Theoretical advantages of this design have been estimated and are effective in the high-Mach number range. The practical realization of this engine concept is critically dependent on the degree of mixing attainable at high velocities and is therefore difficult to assess without long and intensive studies. Some "spin-offs" from this concept can be of importance in more conventional scramjet designs; notably, the interactions between shock waves of various strengths, turbulent mixing, and combustion deserve special detailed study.

Future Plans

The ODWE program was terminated at the end of FY89. The experimental program showed partial success and would have required additional measurements for unambiguous answers. The numerical program was a complete success, with the important validation of the code against previous ballistic experiments. The code has been used as a launch platform for the development of other, more sophisticated codes for nonequilibrium flows. Extensions to multiple-temperature and strongly ionized flows have been done and are being pursued further by one of the co-investigators (JLC).

Publications

1. Adelman, H. G.; Cambier, J. L.; and Menees, G. P. "Experimental and Analytical Investigations of Wave Enhanced Supersonic Combustors." AIAA Paper 89-2787, July 1989.
2. Menees, G. P.; Adelman, H. G.; Cambier, J. L.; and Bowles, J. V. "Wave Combustors for Trans-Atmospheric Vehicles." Presented at the 9th Int. Symposium on Air-Breathing Engines, Greece, Sept. 1989.
3. Menees, G. P.; Adelman, H. G.; and Cambier, J. L. "Analytical & Experimental Validation of the Oblique Detonation Wave Engine Concept." Presented at the 75th AGARD Symposium on Hypersonic Combined Cycle Propulsion, Madrid, Spain, May 1990.



The position of flame/shock decoupling. The color figure shows temperature contours in H_2 -air; $M = 6.46$.

Large-Eddy Simulations of Ramjet Combustion Instability

Suresh Menon, Principal Investigator

Co-investigator: Wen-Huei Jou

QUEST Integrated, Inc.

Research Objective

The objective of this research is to study, through large-eddy simulations, the complex interactions among acoustic waves, vortex motion, and combustion heat release in a ramjet combustor. These interactions often lead to combustion instability in such a device.

Approach

The unsteady compressible Navier-Stokes equations, together with a thin-flame model for a premixed combustion, are solved using MacCormack's explicit finite-volume scheme. The enthalpy in the energy equation contains a heat-of-formation term that can be varied to reflect the effects of the equivalence ratio of a fuel mixture. The turbulent flame speed appears explicitly in the combustion model as a function of the laminar flame speed and the subgrid turbulence kinetic energy.

Accomplishment Description

Large-eddy simulations of premixed combustion in an axisymmetric ramjet combustor were carried out. Both "stable" and "unstable" combustion were simulated numerically. The stable combustion flow field is characterized by low-level (15% of mean pressure) pressure fluctuations and vortex roll-up/pairing processes occurring in the shear layer. The flame is entrained by the vortical structures as it propagates downstream. In the unstable combustion flow field, large-amplitude, low-frequency pressure fluctuations occur at levels

as high as 50% of the mean pressure, and a large, hooked vortex/flame structure propagates through the combustor at the low frequency. The computed flame structure, the phase relation between the pressure and velocity fluctuations, and the fluctuation levels in the combustor for unstable combustion are in remarkable agreement with experimental results (e.g., Smith and Zukoski, AIAA Paper 85-1248). Each simulation of combustion instability in the ramjet using a grid of 256×64 points required around 24 Cray-2 hours and about 5 megawords of memory.

Significance

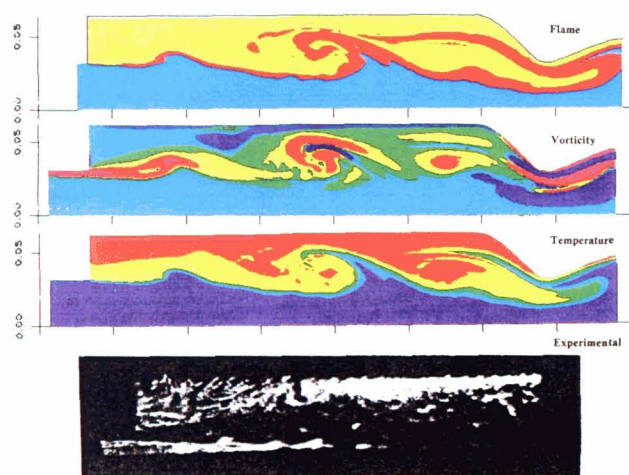
The numerical simulation of combustion instability has provided, for the first time, detailed information on an unstable flow field in a ramjet. This capability will be exploited in a theoretical framework to analyze the results of simulations for practical applications, such as developing strategies for actively controlling the instability.

Future Plans

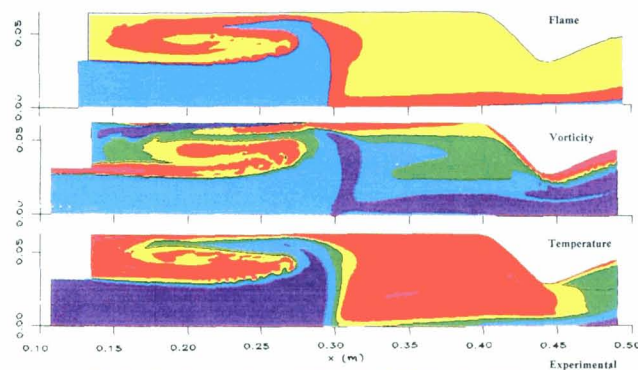
A new research study has been initiated to further improve the large-eddy simulation model and to study numerically active control techniques for controlling ramjet combustion instability.

Publications

Menon, S., and Jou, W.-H. "Large-Eddy Simulation of Combustion Instability in an Axisymmetric Ramjet Combustor." AIAA Paper 90-0267, Jan. 1990.



a. Stable Combustion



b. Unstable Combustion

Typical flow fields for (left) stable and (right) unstable combustion. (Experimental visualization from Smith and Zukoski, AIAA Paper 85-1248.)

Mixing Enhancement for Scramjet Flameholders

Suresh Menon, Principal Investigator
Co-investigator: Emerick Fernando
QUEST Integrated, Inc.

Research Objective

The primary objective of this research is to investigate the unsteady mixing processes between a supersonic air stream and tangentially injected fuel near three-dimensional flameholders. The numerical work is being carried out in parallel with an experimental study of similar flameholder configurations.

Approach

The compressible, three-dimensional Navier-Stokes equations are solved using a finite-volume scheme based on MacCormack's technique. A fully implicit scheme is used for steady-state calculations, and a large-eddy simulation code is being developed for unsteady simulations. To resolve shocks in the flow field, the Roe-averaged flux difference splitting scheme is used.

Accomplishment Description

The research is being carried out in two phases. In the first phase, an explicit three-dimensional version of the code was used to carry out direct simulations of temporally evolving compressible mixing layers, to study the effect of compressibility on secondary instability in the mixing layer. The evolution of three-dimensional modes and the subsequent formation of streamwise vortices for various convective Mach numbers were numerically simulated. For a 64^3 grid resolution, approximately 12 megawords of memory and 4 Cray-2 hours were required for each simulation. In the second phase, a fully implicit version of the code was used to obtain steady-state

turbulent supersonic flows past rearward-facing steps with and without spanwise modification to the flameholder. Using a $74 \times 64 \times 32$ grid and the implicit scheme, steady-state solutions required about 8 Cray-2 hours and 14 megawords of memory. The steady-state solution will be used to initialize the flow field for the unsteady simulations, which are currently being initiated.

Significance

In the Mach number regime from 2 to 4, stability analysis indicates that the most unstable mode is three-dimensional. Enhancing three-dimensional instability may lead to an increase in streamwise vorticity in the mixing layer, which could result in increased fuel/air mixing. Mixing enhancement is essential for improving the combustion efficiency of the scramjet engine.

Future Plans

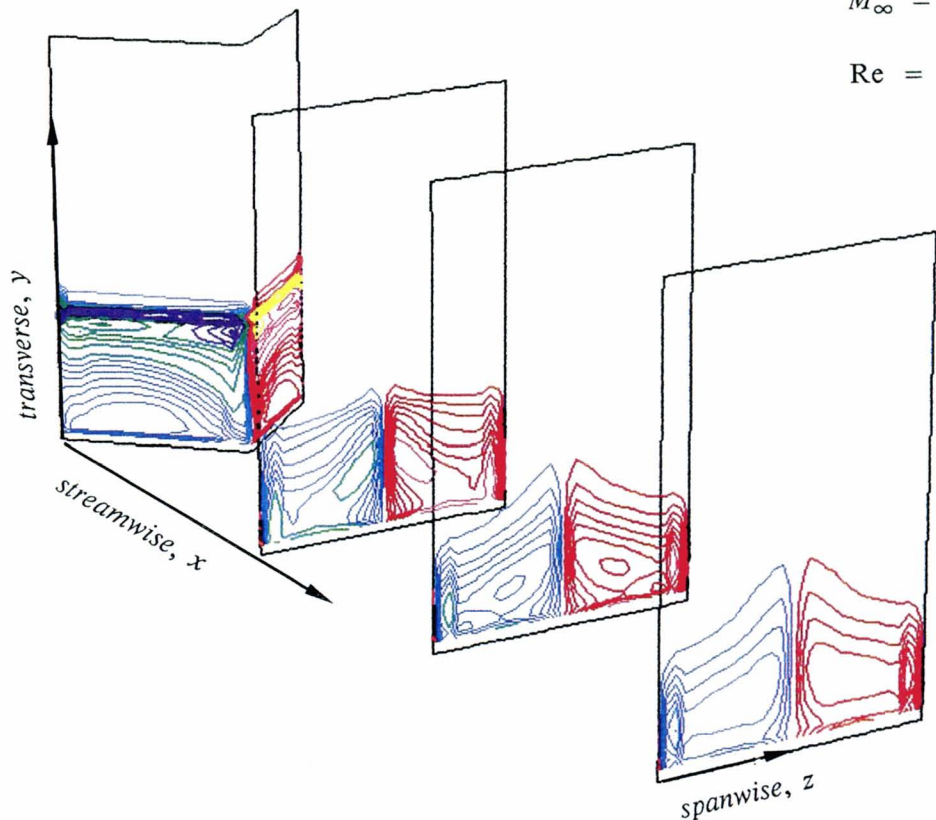
Steady-state solutions for supersonic flow past a flameholder geometry identical to the experimental configuration will be carried out for code validation. Tangential fuel injection near the flameholder will be included. Large-eddy simulations will be carried out to investigate the unsteady aspects of fuel/air mixing near three-dimensional flameholders.

Publications

Menon, S., and Fernando, E. "A Numerical Study of Mixing and Chemical Heat Release in Supersonic Mixing Layers." AIAA Paper 89-0152, Jan. 1990.

CONTOUR LEVELS

-14500.0
-13500.0
-12500.0
-11500.0
-10500.0
-9500.00
-8500.00
-7500.00
-6500.00
-5500.00
-4500.00
-3500.00
-2500.00
-2100.00
-1800.00
-1500.00
-1200.00
-900.000
-600.000
-300.000
300.0000
600.0000
900.0000
1200.000
1500.000
1800.000
2100.000
2500.000
3500.000
4500.000
5500.000
6500.000
7500.000
8500.000
9500.000
10500.00
11500.00
12500.00
13500.00
14500.00



$$M_{\infty} = 3.$$

$$Re = 10^6$$

Streamwise vorticity (Ω_x) downstream of a sawtooth flameholder; $M_{\infty} = 3.0$, $Re = 1 \times 10^6$.

Absorption of Microwave Energy in a Flowing Gas

Charles L. Merkle, Principal Investigator
Co-investigator: S. Venkateswaran
The Pennsylvania State University

Research Objective

To develop a comprehensive computational model to investigate the physics and dynamics of the microwave/gas-dynamic interaction in microwave plasma discharges.

Approach

The full Navier-Stokes equations in axisymmetric form are solved for the gas, coupled simultaneously to Maxwell's equations for the microwave field. An implicit algorithm that had been developed for low-Mach number flows is incorporated in the Navier-Stokes code, and an explicit time-accurate method is used for Maxwell's equations.

Accomplishment Description

A coupled Navier-Stokes/Maxwell solver was developed to model the absorption of microwave energy in helium gas. Flow-field solutions were obtained for various geometries, including the configuration used in the companion experimental program. The effects of several parameters (such as discharge pressure, gas velocity, incident microwave power, and location of a microwave waveguide) on the size, shape, and location of the plasma were investigated. Comparison with experiments showed that except for the threshold powers of the plasma, the trends were predicted accurately. The one discrepancy seemed to be due to the assumption of local thermal equilibrium between electrons and heavy particles in the plasma. Nonequilibrium effects are being included to enhance the present model and improve predictions.

Significance

Microwave propulsion appears to be a promising new concept for space applications such as orbital transfer vehicles. These

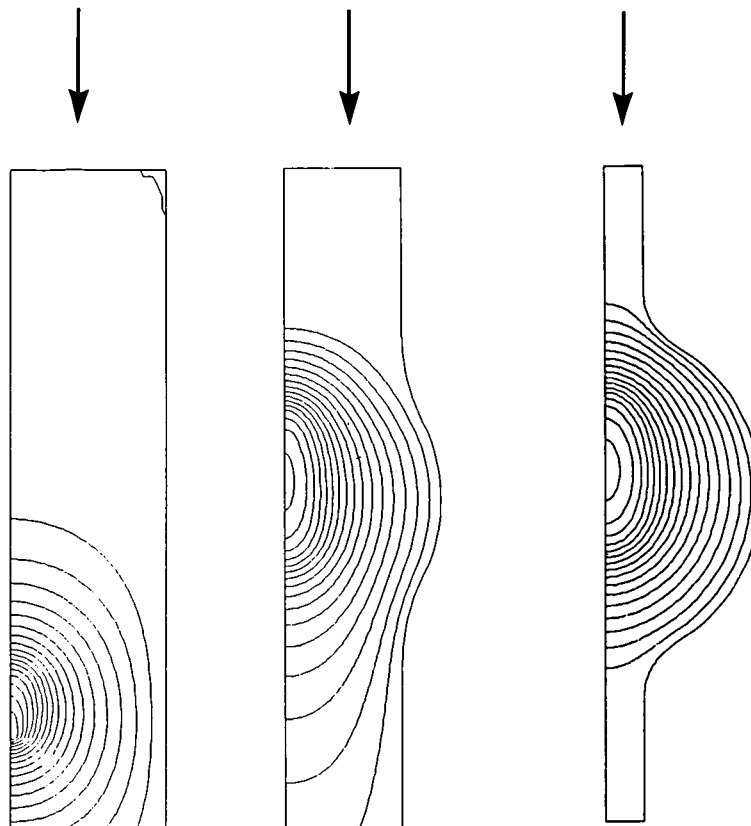
devices provide moderate thrust levels at high specific impulses. Several experimental programs in the country are studying microwave plasma discharges and their application to rocket propulsion. So far, however, theoretical modeling of the problem has received very little attention. Theoretical analysis aided by numerical computations can be very useful in the understanding of the complex interaction between the microwave and gas-dynamic fields.

Future Plans

The present model will be enhanced by the inclusion of nonequilibrium effects. This involves the solution of additional transport equations for the electron number density and electron energy. Validation of the model with experimental data will continue. Future calculations will consider more practical configurations and larger power levels. Further, calculations will be extended to include the rocket nozzle as well as to estimate thrust and overall efficiencies.

Publications

1. Venkateswaran, S.; Merkle, Charles L.; and Micci, Michael M. "Analytical Modeling of Microwave Absorption in a Flowing Gas." Presented at the AIAA 21st Fluid Dynamics Plasmadynamics and Lasers Conference, Seattle, WA, June 1990.
2. Venkateswaran, S., and Merkle, Charles L. "Coupled Navier-Stokes Maxwell Solutions for Microwave Propulsion." Presented at the 12th International Conference on Numerical Methods in Fluid Dynamics, Oxford, England, July 1990.



Temperature contours (drawn at intervals of 500 K) for three different flow geometries: (left) a straight duct, $T_{\text{peak}} = 9335$ K; (center) an intermediate sphere-cylinder, $T_{\text{peak}} = 9060$ K; and (right) the sphere-cylinder used in experiments, $T_{\text{peak}} = 9200$ K.

Numerical Simulation of Advanced Propellers

Christopher J. Miller, Principal Investigator

Co-investigator: Saif A. Warsi

NASA Lewis Research Center/Sverdrup Technology, Inc.

Research Objective

Euler and Navier-Stokes codes for predicting high-speed propfan flow fields have been developed, but need to be verified with experimental data. The objectives for this year involved validation comparisons for both a single-rotation propfan at an off-design condition and a counter-rotation propfan at a design condition. There was also an effort to evaluate a counter-rotation propfan for the highly viscous unsteady flow field at the back of a generic cruise missile body.

Approach

The Euler and Navier-Stokes codes used in this research were developed at the NASA Lewis Research Center by Dr. John Adamczyk. These are finite-volume codes using a four-stage Runge-Kutta integration scheme. An algebraic eddy-viscosity turbulence model is used in the Navier-Stokes code. The wind tunnel test data used for comparison are primarily the overall performance data of a blade row (thrust, power, efficiency). For the counter-rotation propfan at cruise, laser Doppler anemometer data were also compared with the predicted flow field.

Accomplishment Description

Surface static-pressure data are available for a single-rotation, 9-ft-diameter SR-7 propfan that was tested in Modane, France. For the takeoff condition, Euler code predictions capture most of the flow physics, including a leading-edge vortex, and the calculated surface pressures are similar to the test data. The Navier-Stokes code captures the same flow phenomena with fewer mesh points. The calculated surface pressures, although similar to the test data and Euler predictions, show

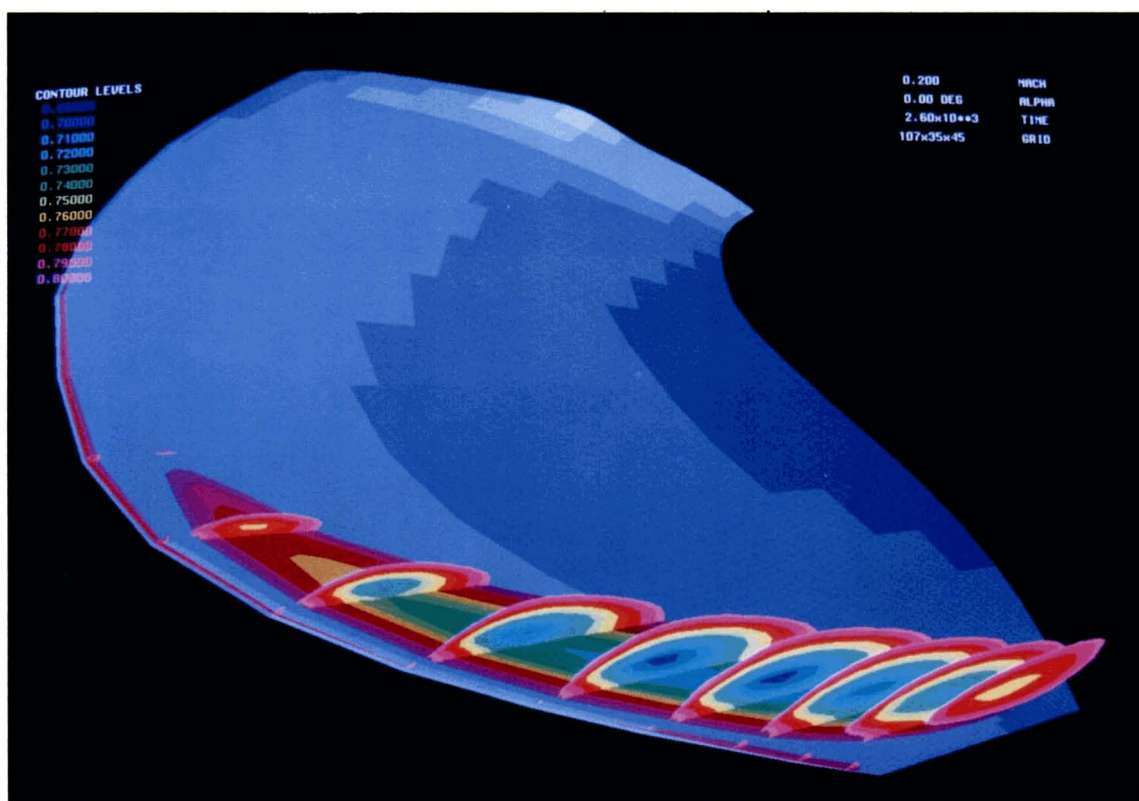
the need for further code improvement. The prediction of the flow field around a counter-rotating propeller for comparison with laser Doppler velocimeter data is not complete. The preliminary results show a general agreement with test data. The Euler predictions do not show the velocity jump across the blade wake that persists in the test data a chord length downstream of the trailing edge. This potential flow feature should be present, and the problem is being investigated. Finally, the Navier-Stokes code was used to predict the performance of a counter-rotation pusher propfan designed to operate on a simulated cruise missile. Because of the thick boundary layer coming off the body and the sharp boat-tail angle ahead of the propeller, this calculation can only be done with a viscous three-dimensional propeller code. There are three basic results: (1) there is a small separation region ahead of the propeller, (2) the propeller accelerates the boundary layer and almost removes the velocity deficit, and (3) the efficiency is roughly 80%. All three results are favorable for the design.

Significance

The validation work has shown that the Euler and Navier-Stokes codes are reliable methods for predicting propfan performance at cruise conditions. For takeoff conditions, the Euler code is qualitatively correct, and the Navier-Stokes code shows promise but needs additional work in the turbulence model.

Future Plans

Continued improvement and validation of the Navier-Stokes code is the highest priority. Both codes will be used to explore new concepts in high-speed propfan propulsion.



Leading-edge vortex, imaged using density, for an LAP SR-7 two-bladed case (#6), computed with the Adamczyk Euler code; 640 cycles, $CFL = -4$, $C_t = 0.1697$, $C_p = 0.2156$, $E_{ta} = 0.6926$, $M = 0.200$, $\alpha = 0.00^\circ$, time = 2.60×10^3 , grid size $107 \times 35 \times 45$.

Quantum Mechanical Reactive Scattering

William H. Miller, Principal Investigator
Co-investigator: John Z. H. Zhang
University of California, Berkeley

Research Objective

To develop new, more efficient methods for carrying out rigorous quantum mechanical reactive-scattering calculations for chemical reactions, and to apply these methods to interesting dynamical problems.

Approach

The coupled-channel Schrödinger equation for reactive scattering (i.e., chemical reactions) is solved by the S-matrix version of the Kohn variational principle.

Accomplishment Description

The S-matrix version of the Kohn variational principle is proving to be an extremely powerful and straightforward way to solve the integro-differential coupled-channel Schrödinger equation for reactive scattering of atoms and molecules. It was shown this year how the method can be readily applied to determine photodissociation and photodetachment cross sections, and also continuum resonance Raman-scattering cross sections. Specific calculations for $\text{H}_2\text{F}^- + h\nu \rightarrow \text{H}_2 + \text{F}$, $\text{HF} + \text{H} + e^-$ were carried out.

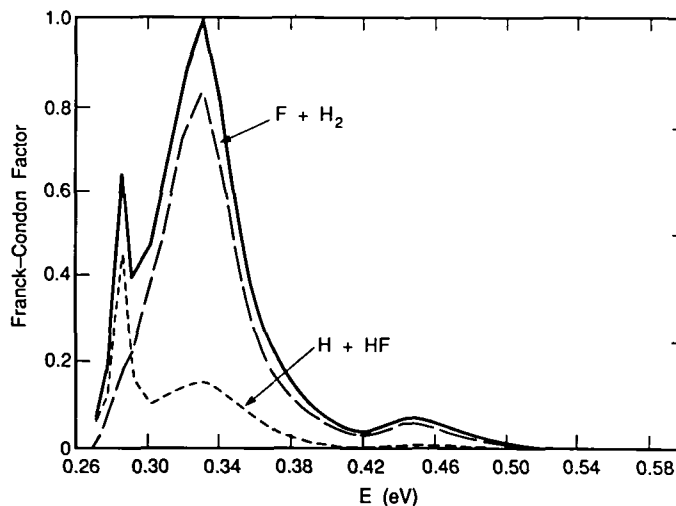
Significance

Quantum reactive scattering provides the most rigorous possible description of a chemical reaction. The present method based on the S-matrix version of the Kohn variational principle is by far the most powerful that has yet been developed for carrying out these calculations. Because of a new generation of experimental studies of these elementary reactions, there is an unprecedented opportunity for detailed comparisons of theory and experiment.

Future Plans

There are a number of opportunities for application of the current method, and there are many avenues for enhancing it to make it applicable to an even wider class of chemical reactions.

Specific plans include the development of various basis set contraction schemes, in order to be able to treat more than three-atom reactive systems.



Photodetachment cross sections (unnormalized) for $\text{H}_2\text{F}^- + h\nu \rightarrow \text{H}_2 + \text{F}$, $\text{HF} + \text{H} + e^-(\epsilon)$. The energy on the horizontal axis is the energy in the neutral scattering system, which is related to the ejected electron energy ϵ by $E = 1.27 \text{ eV} - \epsilon$.

Multi-Tasked Numerical Simulation of Complex Configurations in Hypersonic Flow

Joseph H. Morrison, Principal Investigator
Co-investigator: David L. Whitaker
NASA Langley Research Center

Research Objective

The primary objective of this research is to develop the capability to compute high-speed flows about general hypersonic vehicle configurations. It is necessary to develop the ability to numerically simulate hypersonic flows, both inviscid and viscous, about general configurations to evaluate the design of hypersonic configurations such as that of the National Aero-Space Plane.

Approach

Two- and three-dimensional Navier-Stokes codes, using zonal structured and unstructured meshes, are used to predict high-Mach number flows about hypersonic-flight vehicles. Adaptive grids can easily be generated for the unstructured meshes to improve the accuracy of the calculation.

Accomplishment Description

A two-dimensional flux-difference splitting scheme based on a triangular discretization of the computational domain was developed. The full Navier-Stokes equations can be solved for a laminar flow on a triangular mesh, and an extension of a Cebeci and Smith algebraic turbulence model to the unstructured mesh is under development. The convergence rate of the upwind scheme on an unstructured mesh of triangles, using a pointwise explicit scheme, was found to be very slow. An implicit scheme was implemented using flux-vector splitting to drive the higher order flux-difference split method. Current sparse matrix solution technology was used to obtain, quickly, approximate solutions to the series of sparse linear systems. The figure shows the solution in the P8 inlet using the unstructured solver. Also, a three-dimensional Navier-Stokes zonal structured code was used to solve flows about general

hypersonic configurations and inlets. The code can predict both laminar and turbulent viscous flows using either a time-dependent procedure or a marching procedure. A three-dimensional inlet model required 5 Cray-2 minutes to run in the marching calculation and about 2 Cray-2 hours for a time-dependent solution, using about 5 megawords of storage. Autotasking was used to evaluate the multi-tasking characteristics of the codes. No dedicated processor time was available, so the level of success of these efforts is not known.

Significance

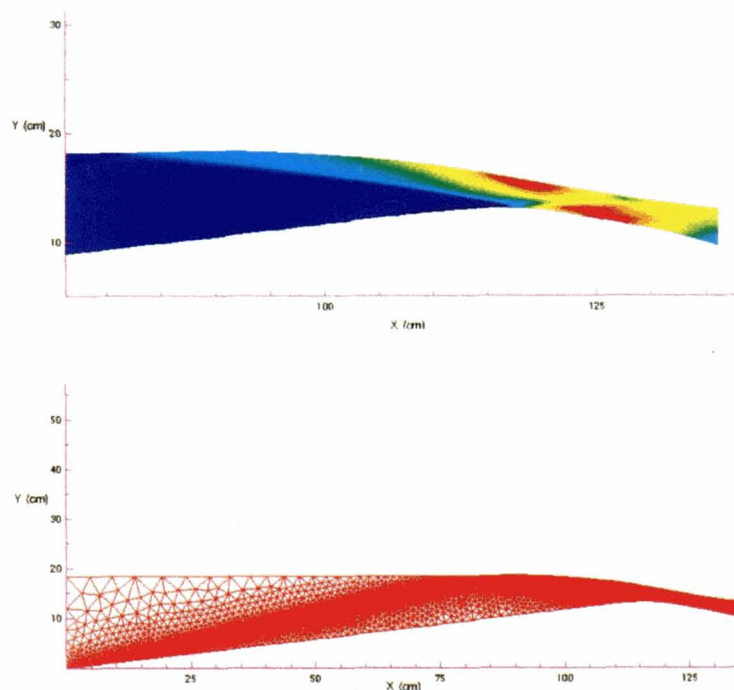
Proposed hypersonic vehicles will operate in flight regimes that are beyond the capability of current wind tunnels to test. Therefore, much of the aerodynamics work will have to be done using numerical simulation. The geometric complexity of the configurations, especially the engines with multiple struts, require either zonal structured grids or unstructured grids.

Future Plans

The unstructured-grid code will be extended to three-dimensional meshes composed of tetrahedrons. The use of grid enhancement through mesh adaptivity will also be explored. Further investigation of implicit methods and the use of sparse matrix solvers will be conducted. The zonal structured code will be extended to include a two-equation turbulence model in order to calculate more accurately the flow field about configurations and inlets in hypersonic flow. The efficiency of the multi-tasking will be evaluated.

Publications

Whitaker, D. L.; Slack, David C.; and Walters, Robert W. "Solution Algorithms for the Two-Dimensional Euler Equations on Unstructured Meshes." AIAA Paper 90-0697, Jan. 1990.



(Top) Pressure contours and (bottom) mesh of triangles for a P8 inlet.

Wall-Bounded Turbulent Flows

Robert D. Moser, Principal Investigator
Co-investigator: John Kim
NASA Ames Research Center

Research Objective

To study the physics of wall-bounded turbulent shear flows, including the effects of flow complications (e.g., curvature), flow control, and coherent structures.

Approach

Direct numerical simulations are performed of turbulent flows in simple geometries (plane and curved channels), and the resulting fields are analyzed.

Accomplishment Description

Three major direct numerical simulations were completed to study three specific questions. First, a high-Reynolds number ($Re_d = 8000$) channel flow was computed to investigate Reynolds number effects. Near-wall statistics were found to be dependent on Reynolds number, in agreement with boundary layer computations of Spalart. These results are being used to develop improved turbulence models. Second, a plane channel with active control was simulated, and control methods

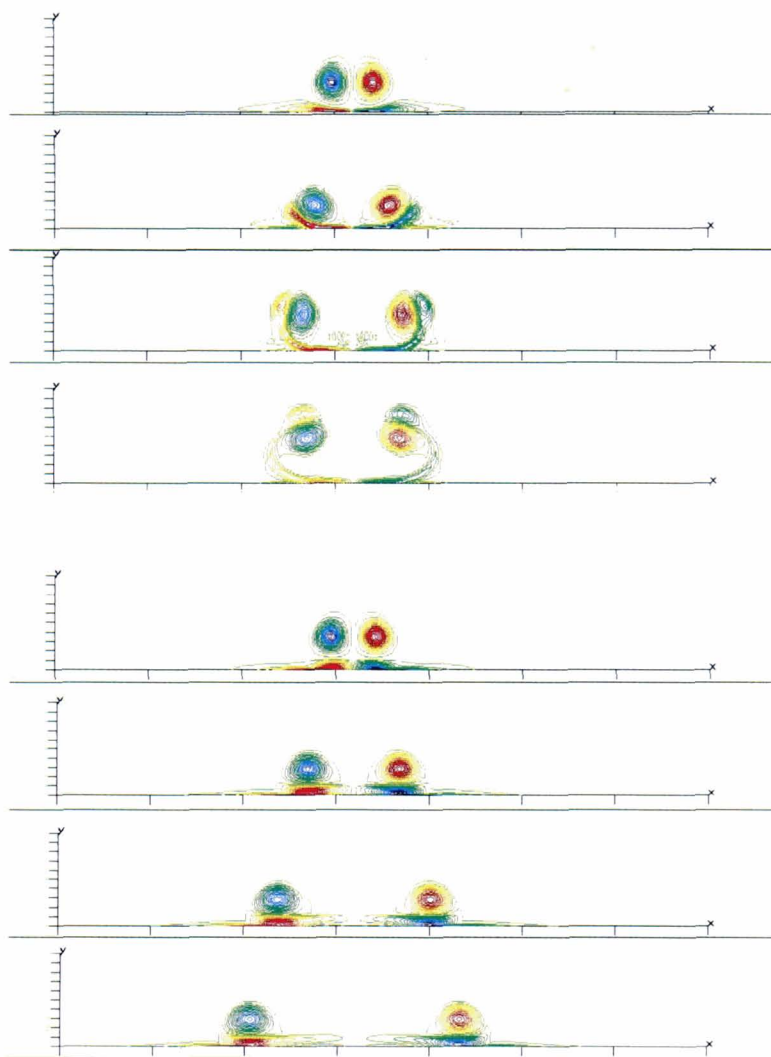
were improved by the use of spanwise and/or normal velocity control at the wall, resulting in a 30% reduction in drag. The mechanisms of this drag reduction are being investigated. Finally, a strongly curved turbulent channel was computed. It was found that large-scale Gortler vortices are not two-dimensional as they are with weak curvature.

Significance

The numerical simulations allow access to information about the flow that is not available by other means. Both curvature and turbulent drag reduction are of great technological importance in aeronautics and other applications.

Future Plans

The simulations are being analyzed. It is expected that the curvature simulations will be used to evaluate and develop turbulence models that include curvature effects, and future simulations with more realistic and improved drag reduction algorithms are being planned.



Vortex pair impinging on a wall without control (top) and with control (bottom). Note that with-control vortices behave as on a wall with slip.

Electronic Structure of Superconductors

Wolfgang Mueller, Principal Investigator
Analatom, Inc./NASA Lewis Research Center

Research Objective

To investigate the electronic structure of high-temperature superconductors with the quantum chemical cluster approach, perform theoretical analyses for critical materials properties, and contribute to a better understanding of the mechanism responsible for high-temperature superconductivity.

Approach

Large-cluster calculations are carried out using the self-consistent, relativistic scattered-wave method, which is based on the Dirac wave equation. The method uses the multiple scattering technique and Slater's statistical approximation for electron exchange and correlation.

Accomplishment Description

Fully relativistic calculations have been performed for Y-Ba-Cu-O using two different clusters: $\text{Y}_2\text{Ba}_2\text{Cu}_{12}\text{O}_{18}$ with three Cu-O layers, and $\text{YBa}_2\text{Cu}_6\text{O}_{20}$ with four Cu-O layers. The accompanying figure shows the valence and conduction electronic structure per $\text{YBa}_2\text{Cu}_3\text{O}_7$ cell as derived from the two clusters, and a decomposition into atomic contributions. The prominent double peak just below the Fermi level, and other features that are observed in photoemission experiments, are clearly resolved. For $\text{Y}_2\text{Ba}_2\text{Cu}_{12}\text{O}_{18}$, final-state photoemission relaxation contributions have been calculated

for about 25 valence and core states. Final-state effects (relative to the cluster Fermi level) are found to be small within the valence band, but significantly affect the core levels, where the remaining errors are 1 to 5 eV. A typical calculation requires about 5 Cray-2 hours and 2 megawords of memory.

Significance

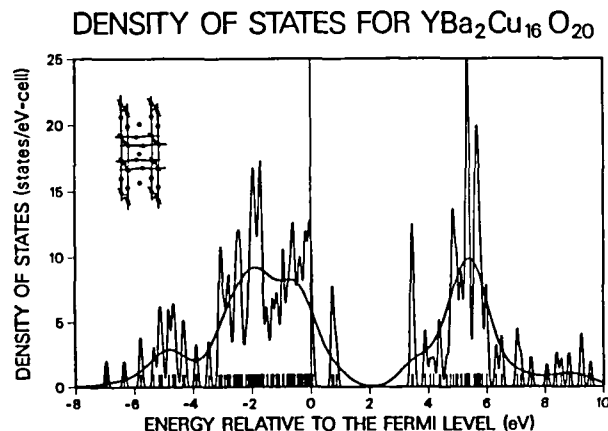
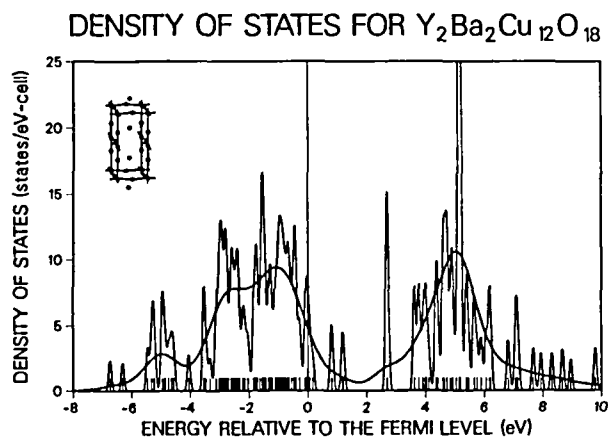
The investigation of spin-orbit, relaxation, and correlation effects in cluster calculations is important for understanding the discrepancies between band structure results and photoemission data. Large clusters are shown to provide a good description of the electronic structure of these materials.

Future Plans

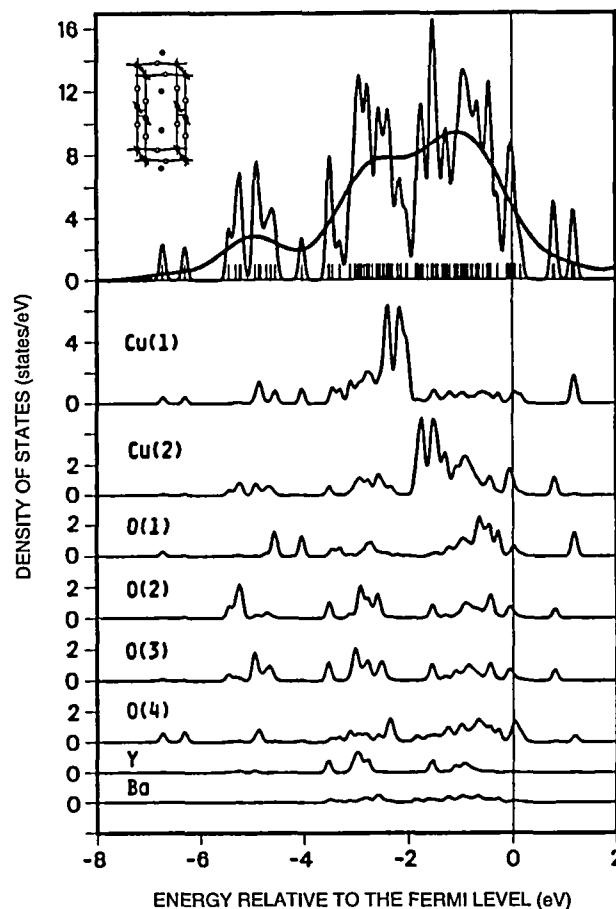
Extended cluster and plane-wave calculations will be performed for thin films of Y-Ba-Cu-O on various substrates.

Publications

1. Mueller, W. "Relativistic Cluster Results for $\text{YBa}_2\text{Cu}_3\text{O}_7$." *Physica C* 162-164 (1989): 1357.
2. Mueller, W. "Valence and Core Electronic Structure of the 90K Superconductor Y-Ba-Cu-O." Presented at the 10th Canadian Symposium on Theoretical Chemistry, Banff, Alberta, Aug. 1989.



Density of states for (top) $\text{Y}_2\text{Ba}_2\text{Cu}_{12}\text{O}_{18}$ and (bottom) $\text{YBa}_2\text{Cu}_6\text{O}_{20}$.



Atomic contributions to the density of states.

Transonic Potential Flow about Transport Aircraft

D. A. Naik, Principal Investigator

Co-investigators: A. M. Ingraldi, R. G. Jorstad, T. A. Reyhner, and S. F. Yaros
ViGYAN, Inc./NASA Langley Research Center/The Boeing Company

Research Objective

To analyze transonic flow about different transport airplane configurations.

Approach

A transonic full-potential code was used in the analysis. Considerable flexibility is possible in the choice of the airplane configuration because the code uses Cartesian or cylindrical grids that intersect, rather than fit, the complex body. The code uses a multigrid scheme to accelerate convergence and is vectorized for optimum performance on a Cray-2 supercomputer. An unstructured-grid post-processing code was used to analyze flow solver output.

Accomplishment Description

Three different airplane configurations were analyzed: (1) a generic high-wing transport, (2) a generic low-wing transport, and (3) a High Speed Civil Transport (HSCT). The first two transports are based on existing wind-tunnel models that are being experimentally evaluated in the NASA Langley 16-ft

transonic tunnel. The accompanying figure shows upper- and lower-surface Mach contours for the HSCT when it is operating in the transonic regime. The original HSCT geometry is shown in silhouette. This analysis involved 18,000 surface intersections within a Cartesian mesh of 4 million grid points. Four levels of multigrid cycling were used. This computation took 843 CPU seconds and 12 megawords of memory on the Cray-2. A total of 17 hours of CPU time was used during the 1989-90 operational period.

Significance

This work is part of an ongoing study at NASA Langley Research Center about the optimum reduction of the adverse installation effects of the propulsion systems of transport aircraft.

Future Plans

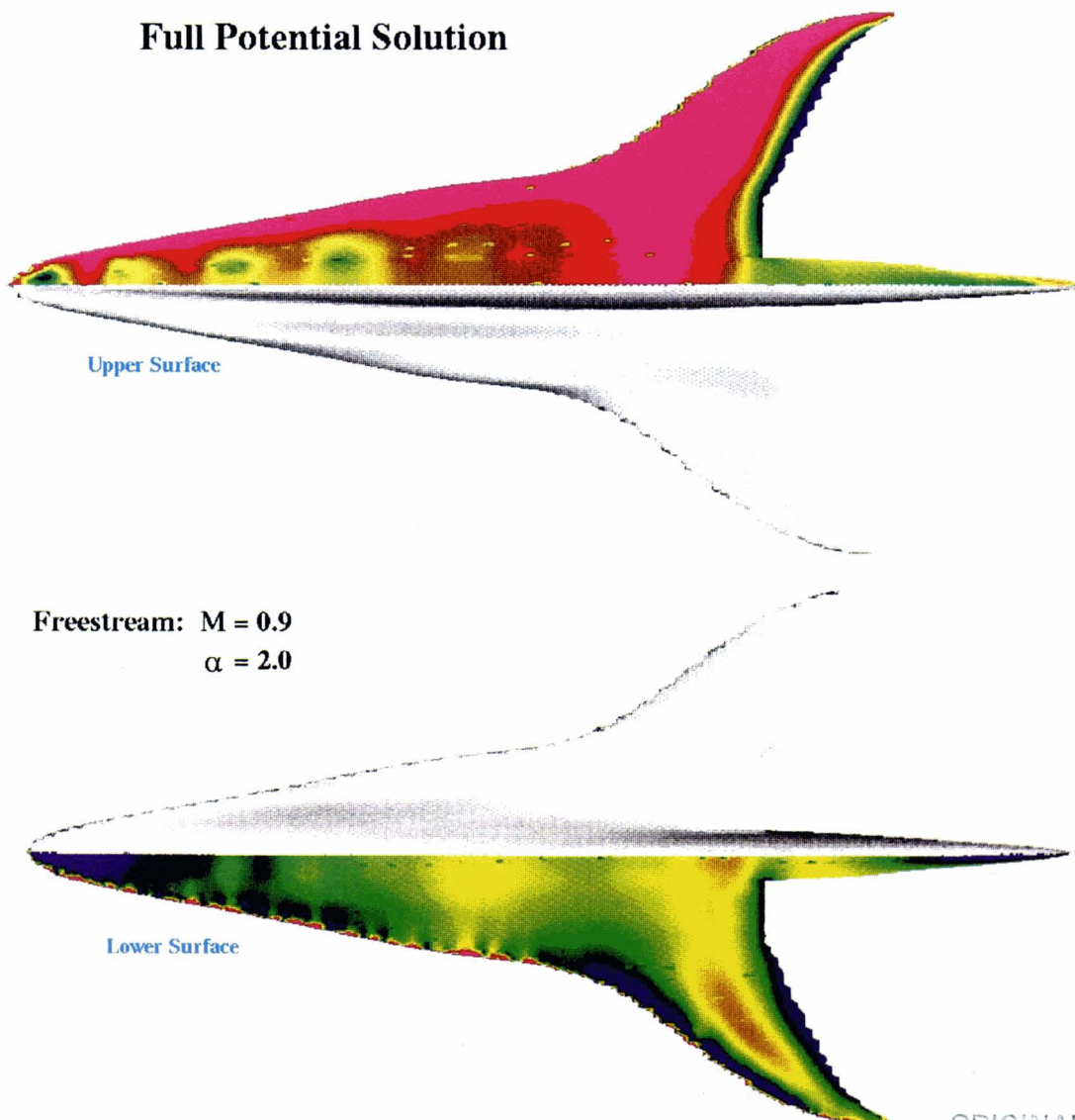
This project has been moved to the supercomputer network system at the NASA Langley central-site computers.

Full Potential Solution

Mach

9.459E-01
9.431E-01
9.403E-01
9.375E-01
9.347E-01
9.319E-01
9.291E-01
9.263E-01
9.234E-01
9.206E-01
9.178E-01
9.150E-01
9.122E-01
9.094E-01
9.066E-01
9.038E-01
9.009E-01
0.981E-01
0.953E-01
0.925E-01
0.897E-01
0.869E-01
0.841E-01
0.813E-01
0.784E-01
0.756E-01
0.728E-01
0.700E-01

Freestream: $M = 0.9$
 $\alpha = 2.0$



ORIGINAL PAGE
COLOR PHOTOGRAPH

Full-potential solution for the Mach contours for the High Speed Civil Transport; $M = 0.9$, $\alpha = 2.0^\circ$.

Unsteady Flow Field on an Advanced Propeller

R. M. Nallasamy, Principal Investigator
Sverdrup Technology, Inc./NASA Lewis Research Center

Research Objective

The objective of this investigation is to understand the aerodynamics and acoustics of an advanced propeller at an angle of attack. A detailed numerical study of the unsteady flow field of the propeller at different operating conditions (cruise and takeoff) is attempted.

Approach

Unsteady three-dimensional Euler equations are solved for the case of an advanced propeller at an angle of attack. The flow field is represented by multiblock composite grids. The solution at each time step is updated by having only one block in memory while other blocks are stored in solid-state storage devices. Fine-grid representation of the blade surface and regions near the blade surface is used to capture the leading-edge vortex.

Accomplishment Description

The three-dimensional Euler code capable of handling unsteady flow fields, developed by Prof. Whitfield of Mississippi State University, was used. The flow field was represented by multiblock composite grids. Solutions were obtained for cruise conditions of the eight-blade flight test configuration. A fine-grid solution was obtained for takeoff conditions of the two-blade wind tunnel test configuration. This solution indicated the presence of a leading-edge vortex on the blade throughout the revolution of the propeller. The size, strength, and chordwise location of the vortex depended on

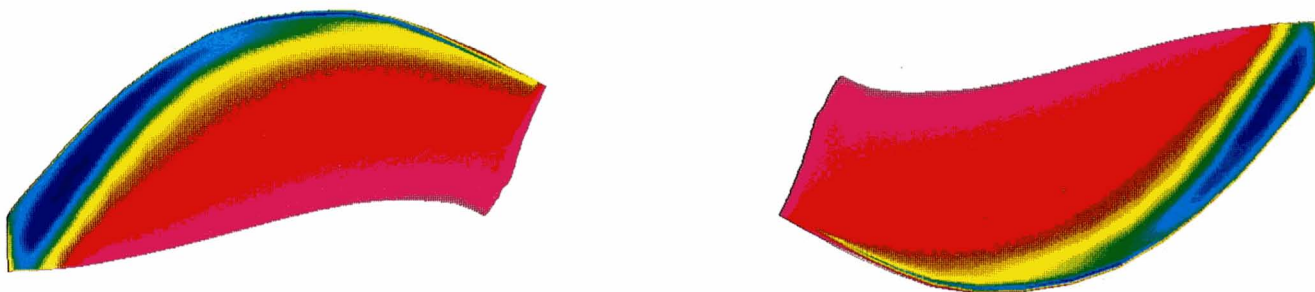
the azimuthal position of the blade. The accompanying figure shows the pressure contours indicating the leading-edge vortex on the suction surfaces of the two blades. Note the difference in size of the vortex between the two blades—the vortex is larger on the blade going down. The multiblock composite grid computations use 4 megawords of memory and 12 hours on the Cray Y-MP for one revolution of the propeller.

Significance

The cruise-condition solutions of the eight-blade configuration, indicate that significant azimuthal variations of blade loading occur with changes in the angle of attack. A shock appearing near the trailing edge on the suction surface of the blade extends across the passage to the pressure surface of the successive blade during the highly loaded part of the revolution. Such details of shocks and shock motion are of great importance in understanding the advanced propeller performance and acoustics.

Future Plans

The unsteady solution will be obtained for different operating conditions and Mach numbers in order to make detailed comparisons with test data. A higher spatial resolution throughout the flow field will facilitate the direct computation of acoustic levels from the flow solutions.



Pressure contours indicating the leading-edge vortex on the blades of a two-blade wind tunnel test configuration.

Pegasus™ Aerodynamic Analyses

David Nixon, Principal Investigator

Co-investigators: Gary D. Kuhn, Steven Caruso, and Michael R. Mendenhall
Nielsen Engineering & Research, Inc.

Research Objective

The objective of this work is the prediction of hypersonic aerodynamic and thermodynamic characteristics of Pegasus™, the world's first air-launched satellite launch vehicle. Of particular interest is the wing-fuselage-fairing region, where the flow may be dominated by shock-wave/boundary-layer interactions, and the tail region, where the control effectiveness may be influenced by the wing wake and forebody vorticity.

Approach

Three-dimensional Navier-Stokes codes are used to predict the flow characteristics around the Pegasus wing, fuselage, and tail for a range of Mach numbers and angles of attack that correspond to a typical flight profile.

Accomplishment Description

In the previous year, the NASA Ames F3D code was used to predict the flow characteristics of the Pegasus wing and fuselage. In the current year, calculations were extended to include the fuselage-tail region, using the Chimera composite-grid scheme developed at NASA Ames to couple the tail grid with the wing-fuselage-fairing grid. The results provided information on the complexity of the flow field in the wing-fuselage-fairing region and in the tail region. The results for $M = 5$, $\alpha = 5^\circ$ indicate that the wing leading-edge shock wave does not impinge directly onto the fuselage or the fairing for this flight condition because of the complicated interaction with the flow around the fuselage. The results also indicate flow separation under the wing and over the wing-fuselage-fairing aft of the wing trailing edge. This separation requires further study to determine how it may change with different flight conditions. The aerodynamic characteristics from these predictions are in good agreement with simpler engineering

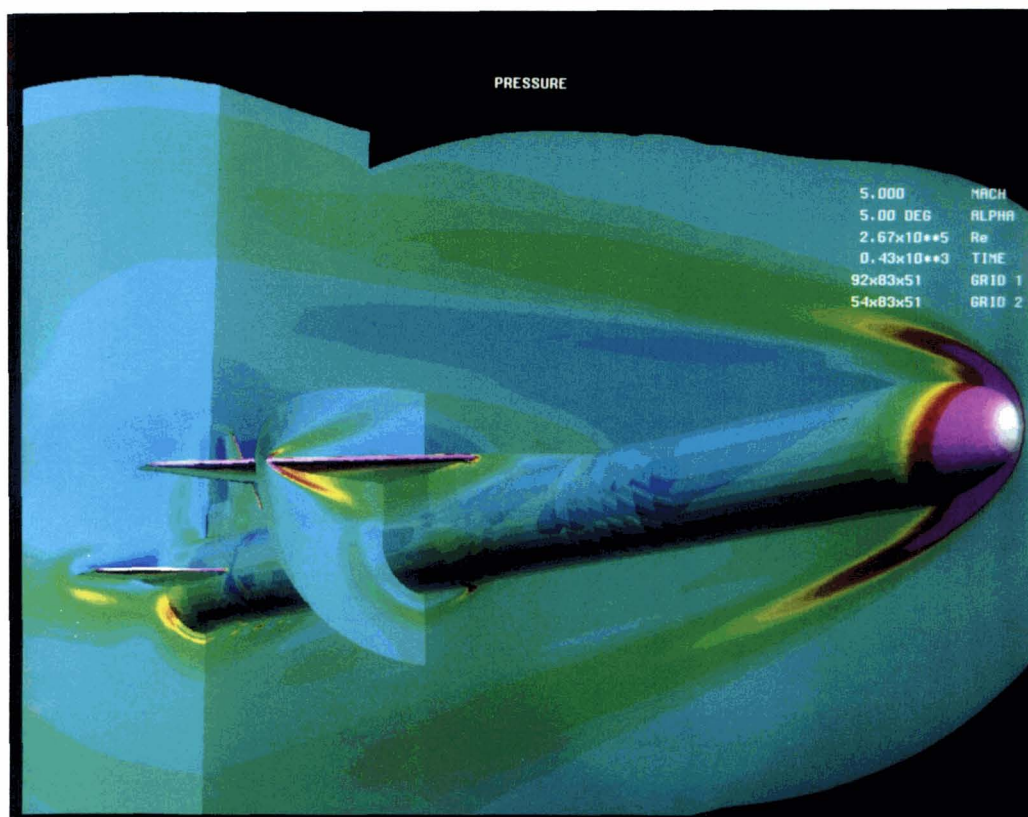
methods, further verifying the predicted aerodynamic loads used for the preliminary design analyses. A typical calculation of the full configuration required about 35 Cray-2 hours and 15 megawords of memory.

Significance

Pegasus, a winged vehicle with flexible operations permitting launch into orbit, NASP trajectories, and ICBM trajectories is the first major hypersonic vehicle to go from concept to flight with its design based solely on computational fluid dynamics (CFD), without wind tunnel tests. Aerodynamic characteristics for flight control and trajectory optimization are available from standard codes, but several complex problems, critical to successful vehicle design and operation, are beyond the capability of these codes. Results from the 1989-90 effort on NAS have demonstrated that available CFD codes have the ability to predict the fluid dynamic properties in complex three-dimensional flow regimes to assist in design analyses of new flight vehicles.

Future Plans

The aerodynamic characteristics of the Pegasus full configuration will be predicted at three or four additional flow conditions in a typical flight trajectory, to complement the results obtained in 1989 ($M = 5$, $\alpha = 5^\circ$). These results will identify locations of possible high heating rates and possible loss of control effectiveness and will provide guidance for modification of the thermal protection and control systems. The grid resolution in the wing-fuselage-fairing region will be refined to increase the accuracy of the heat transfer calculations. Predicted local heat-transfer rates on the fairing and wing will be compared with flight test data from the first Pegasus flight.



Pressure contours for Pegasus.

CRITCHFIELD
COLLECTOR PHOTOGRAPH

Navier-Stokes Simulation of Separated Turbulent Flow around a Generic Fighter Aircraft

Charles R. Olling, Principal Investigator

Co-investigators: Pradeep Raj and Josef S. Sikora

Lockheed Aeronautical Systems Company

Research Objective

To numerically simulate separated turbulent flow around a complete generic fighter aircraft with powered nacelles at high angles of attack, and to investigate the applicability of different turbulence models to this problem.

Approach

The three-dimensional Reynolds-averaged Navier-Stokes equations are solved with a finite-volume Runge-Kutta time-stepping method, TRANSAM.

Accomplishment Description

Flow was computed around a supersonic V/STOL fighter aircraft with powered nacelles. The free-stream Mach number was 1.19 and the angle of attack was 15° . A 27-zone grid with 378,000 cells was generated by a method optimizing a measure of uniformity and orthogonality. The full Navier-Stokes equations with the Chien two-equation $k-\epsilon$ turbulence model were solved in the zones surrounding the wing, and the Euler equations were solved elsewhere. About 15 points were located inside the boundary layer. The results were compared with those obtained using the Baldwin-Lomax turbulence model on the same grid. The figure illustrates the surface pressure coefficient. High values are blue and low values are magenta. A total of about 20 Cray-2 hours and about 17 megawords of memory were used for an average case. Grid points were added in the streamwise direction toward

the wing trailing edge and in the spanwise direction toward the wing root and tip. The resulting grid had a total of 434,924 cells and 36 zones. The full Navier-Stokes equations with the Baldwin-Lomax turbulence model as modified by Degani and Schiff were solved in the zones surrounding the wing, and the Euler equations were solved elsewhere. The results were compared with those previously obtained using the Baldwin-Lomax model without the Degani-Schiff modification on a coarser grid.

Significance

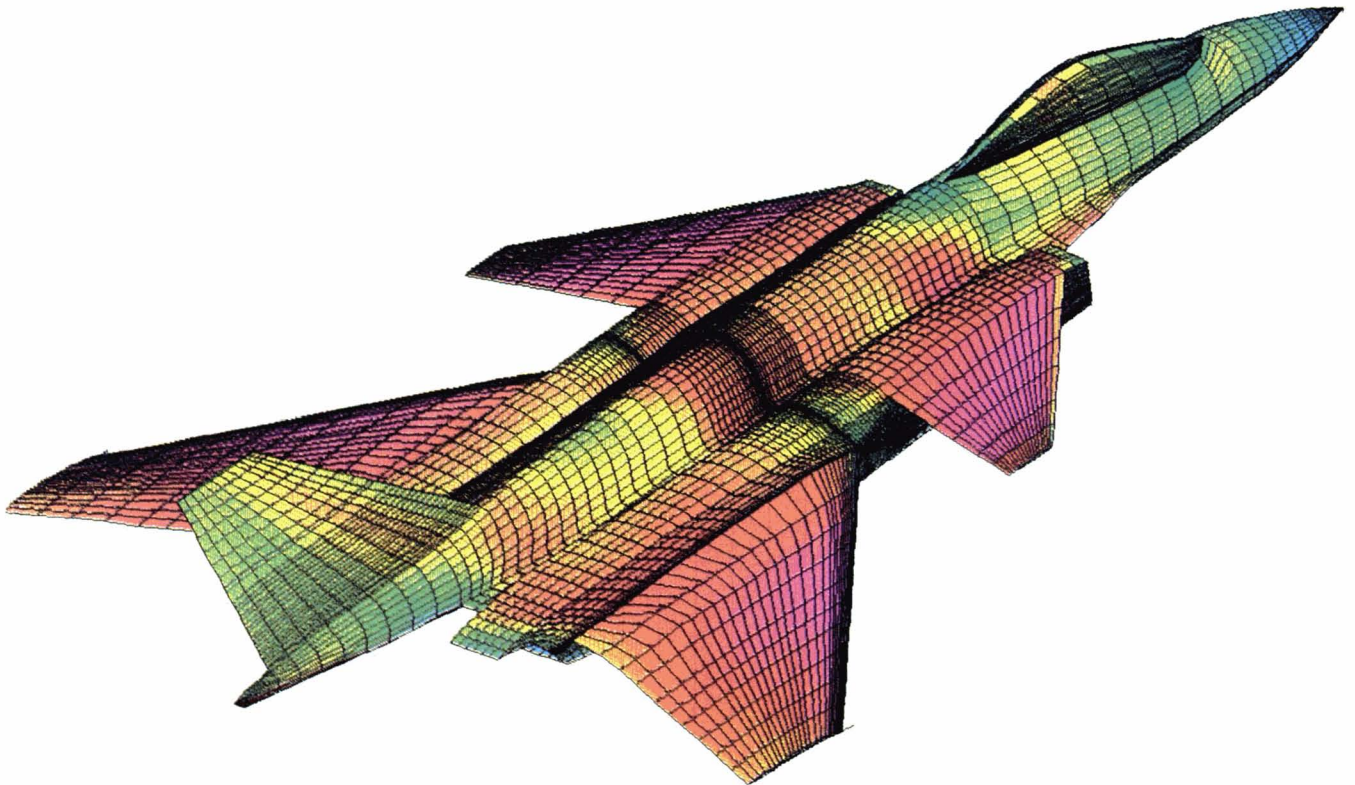
The study is applicable to the design of advanced fighter aircraft.

Future Plans

A grid will be generated for the complete F/A-18 aircraft with clustering suitable for viscous computations near all solid surfaces. Navier-Stokes computations will be performed with the Johnson-King turbulence model on the wing and the Baldwin-Lomax turbulence model with the Degani-Schiff modification elsewhere.

Publications

Olling, C. R.; Raj, P.; and Miranda, L. R. "Aerodynamic Analysis Using Euler/Navier-Stokes Equations." To be published in *Advances in Computational Fluid Dynamics*, ed. W. G. Habashi and M. M. Hafez, 1990.



Computed surface pressures for a supersonic V/STOL fighter aircraft with powered nacelles.

Navier-Stokes Simulation of Turbulent Flow over Rectangular Cavities Containing Stores

Charles R. Olling, Principal Investigator

Co-investigators: Pradeep Raj and James E. Brennan
Lockheed Aeronautical Systems Company

Research Objective

To numerically simulate turbulent flow over three-dimensional rectangular cavities containing generic missile stores, and to investigate the applicability of different turbulence models to this problem.

Approach

The three-dimensional Reynolds-averaged Navier-Stokes equations are solved with a finite-volume Runge-Kutta time-stepping method, TRANSAM.

Accomplishment Description

Time-dependent flow over an empty cavity was computed with the Chien two-equation $k-\epsilon$ turbulence model. The free-stream Mach number was 1.5. For the cavity, the length-to-depth ratio L/D was 5.1, and the width-to-depth ratio W/D was 2.6. The grid had a total of about 169,000 cells and was broken up into two zones. Time-averaged pressures along the centerline of the cavity floor were compared with the experimental values. The figure shows the evolution of the pressure coefficient in the longitudinal symmetry plane during the time interval 5.33 msec to 6.23 msec. A total of about 85 Cray-2 hours and about 7 megawords of memory were used. A grid was generated for an AIM-9C-SF store model inside a three-dimensional cavity with $L/D = 4.5$ and a free-stream Mach

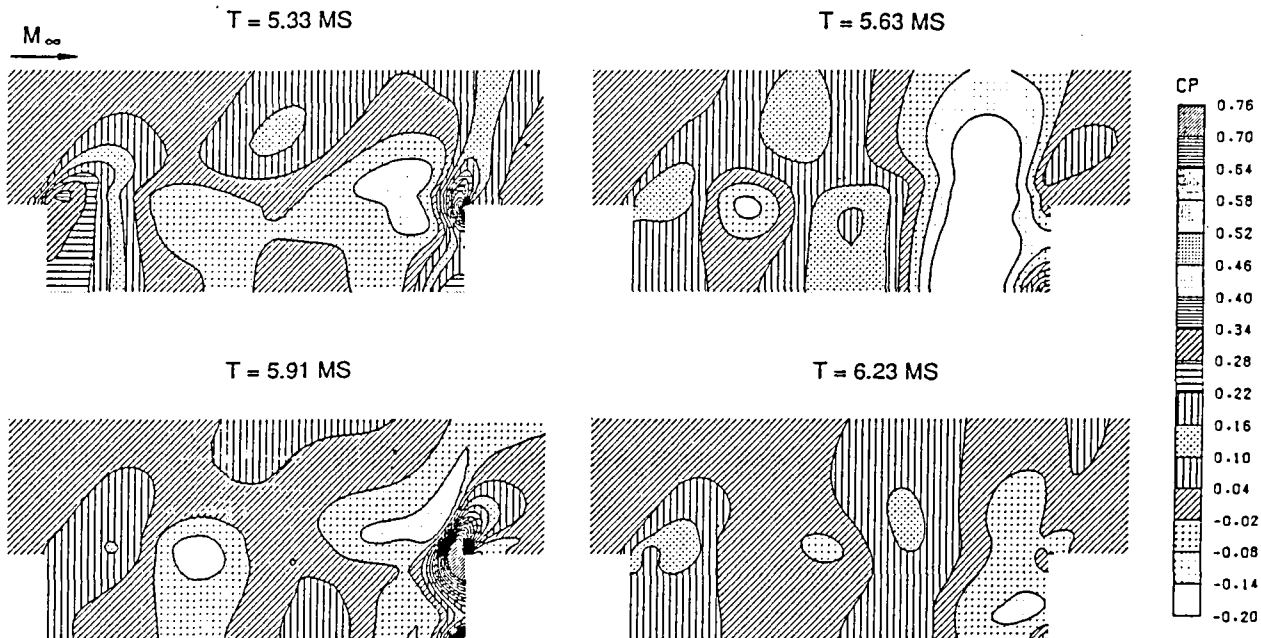
number of 0.95. The grid had a total of about 370,000 cells and was broken up into 40 zones. An H-H-type grid was used outside the cavity and an O-O-type grid around the missile inside the cavity. The grid had a clustering suitable for viscous computations toward all solid surfaces. Time-dependent computations with a CFL number of 1.7 were carried out, requiring about 4 megawords of memory. The Baldwin-Lomax turbulence model with the Degani-Schiff modification was used. The flow was initialized to be stagnant inside the cavity. A total of 10,550 time steps were completed (corresponding to a time of 0.158 msec) at the end of the operational year.

Significance

The study is applicable to understanding supersonic flow over weapon bay cavities.

Publications

1. Olling, C. R. "Navier-Stokes Computations of Turbulent Flow over a Rectangular Cavity." Presented at 1989 SAE Aerospace Technology Conference, Anaheim, Sept. 1989.
2. Olling, C. R.; Raj, P.; and Miranda, L. R. "Aerodynamic Analysis Using Euler/Navier-Stokes Equations." To be published in *Advances in Computational Fluid Dynamics*, ed. W. G. Habashi and M. M. Hafez, 1990.



Pressure distribution at the cavity symmetry plane.

Unsteady Diffusion Flames

Elaine S. Oran, Principal Investigator

Co-investigator: Janet L. Ellzey

Naval Research Laboratory/Berkeley Research Associates

Research Objective

Experiments on jet diffusion flames have shown that two types of instabilities form in the flow field. High-frequency instabilities form between the high-velocity and the low-velocity streams. Low-frequency instabilities form outside the flame zone. The purpose of this study is to investigate the importance of various physical processes to the formation and evolution of these structures.

Approach

The two-dimensional representation of the compressible Navier-Stokes equations with chemical reaction is used to simulate fluctuating H_2-N_2 diffusion flames. The convection terms are solved using the Barely Implicit Correction to Flux-Corrected Transport (BIC-FCT) algorithm. The effects of conductivity, mass diffusivity, and viscosity are included through explicit finite-difference formulations.

Accomplishment Description

An initial study of a confined, steady, laminar flame (Burke-Schumann flame) has shown that the computations agree with the theoretical predictions for flame location. In addition, computations of diffusion flames at velocities of 10 m/sec have shown the effects of viscosity, heat release, and gravity on the two types of instabilities. These results indicate that viscosity and heat release dampen the high-frequency instabilities. In addition, the computations show that low-frequency instabilities in the low-velocity region outside the flame zone are induced when gravity is included in the calculations. These computations require from 4 to 10 megawords of

memory and 10 to 30 hours of CPU time, depending on the resolution.

Significance

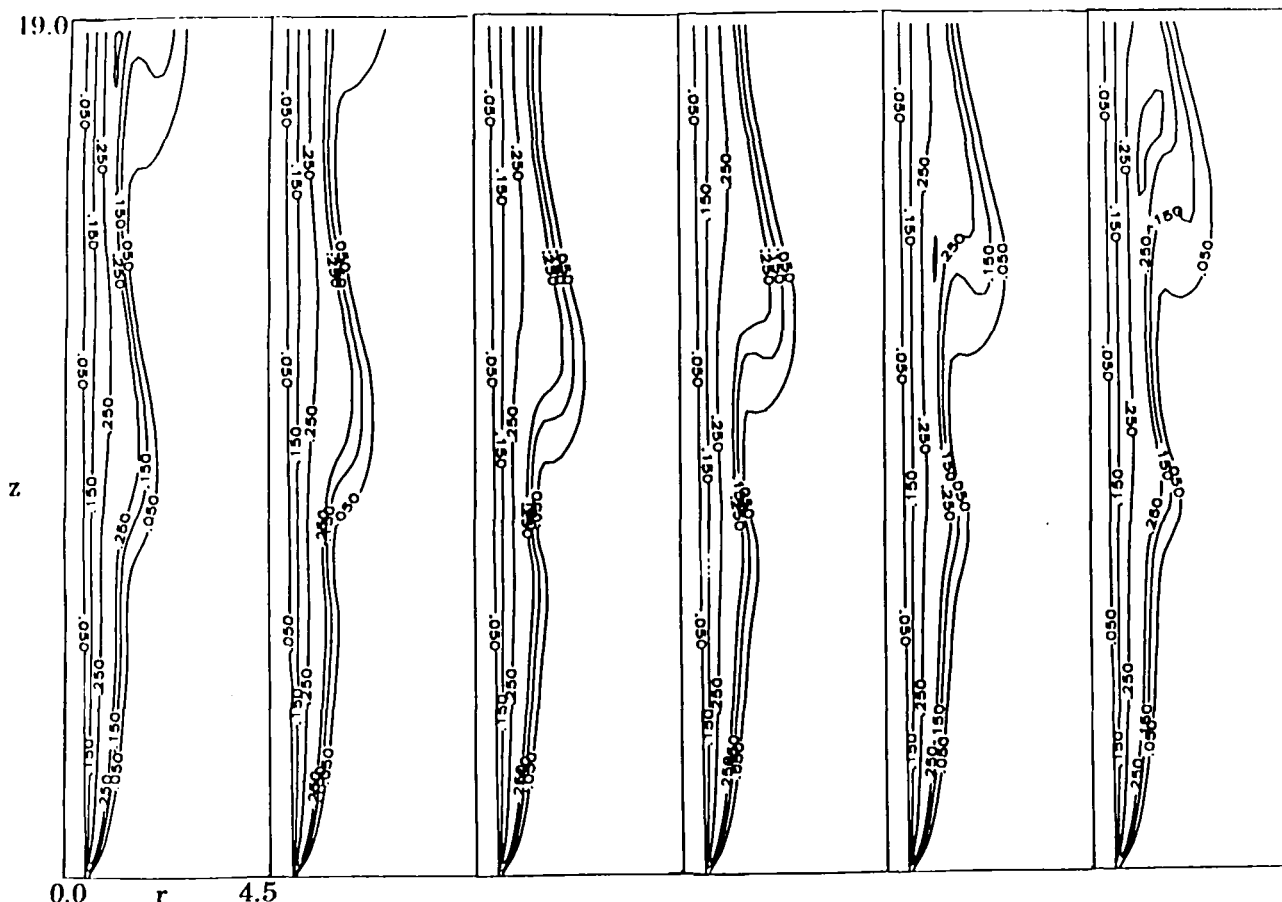
Although numerous experiments on jet diffusion have been conducted, highly resolved time-dependent experimental data on reacting flows are often difficult to obtain. Computations provide detailed data; this is not practical in an experiment. In addition, the effects of individual processes can be examined separately. From the computations at NAS, the effects of heat release, viscosity, and gravity have been examined individually.

Future Plans

The computations will be compared with experiments on jet diffusion flames to further validate the approach. In addition, the importance of multistep chemistry will be investigated.

Publications

1. "Dynamics of Unsteady Diffusion Flames: Effects of Heat Release and Viscosity." Presented at the 12th International Colloquium on the Dynamics of Explosions and Reactive Systems, Ann Arbor, MI, July 1989.
2. "Effects of Heat Release on a H_2-N_2 Diffusion Flame." Presented at the 42nd Annual Meeting of the Division of Fluid Dynamics of the American Physical Society, Palo Alto, CA, Nov. 1989.
3. "Effects of Heat Release and Gravity on an Unsteady Diffusion Flame." Presented at the 23rd International Symposium on Combustion, Orleans, France, July 1990.



Contours of H_2O mole fraction for a hydrogen diffusion flame. This figure shows the formation of low-frequency structures induced by the gravitational field. The time interval between each pair of frames is 11.25 msec.

Flux-Based Finite-Element Formulation Reduces CPU Time for Thermal-Structural Analysis

Ajay K. Pandey, Principal Investigator
Lockheed Engineering and Sciences Company/NASA Langley Research Center

Research Objective

To improve the efficiency and accuracy of finite-element thermal-structural analysis using a flux-based formulation.

Approach

The thermal-structural response of structures is obtained by solving the heat transfer and quasi-static structural equilibrium equations, subject to specified boundary conditions. These equations are written in the conservation form in terms of the conservation variable u and the flux components E , F , and G , as follows: $\partial\{u\}/\partial t + \partial\{E\}/\partial x + \partial\{F\}/\partial y + \partial\{G\}/\partial z = H$. By expressing these equations in the conservation form, the approach provides simplicity in handling different types of boundary conditions, such as applied aerodynamic heat flux and radiation flux. To simplify the nonlinearities of these equations, the fluxes are interpolated linearly over an element. The resulting finite-element equations are in the same form as those obtained from the conventional finite-element technique, as follows: $[K]\{u\} = \{R\}$, where $[K]$ is the finite element stiffness matrix, $\{u\}$ is the vector of nodal unknowns, and $\{R\}$ is the vector of nodal forces. The stiffness matrix $[K]$ obtained from the flux-based formulation is different from that obtained from the conventional approach and can be evaluated in closed form; i.e., numerical integration is not required. A CPU time saving of 80% is achieved in evaluating one three-dimensional stiffness matrix using the flux-based formulation rather than the conventional finite-element formulation. To validate the flux-based finite-element formulation, thermal and structural analyses are performed of a three-dimensional leading-edge

model subjected to a prescribed aerodynamic loading from Mach 5 flow (shown in the upper right figure). The predicted axial stresses are shown on the deformed shape of the leading edge in the lower right figure. The results obtained from the flux-based approach are in excellent agreement with those predicted by the conventional finite-element approach, but have significant CPU time savings.

Accomplishment Description

A thermal-structural analysis approach was developed using the flux-based formulation. The thermal-structural analysis of a three-dimensional leading-edge model subjected to aerodynamic loading was performed and compared with the conventional finite-element solution.

Significance

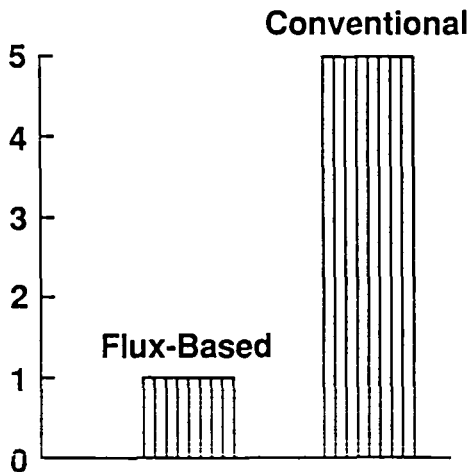
The flux-based formulation yields comparable solution accuracy but provides significant CPU time savings.

Future Plans

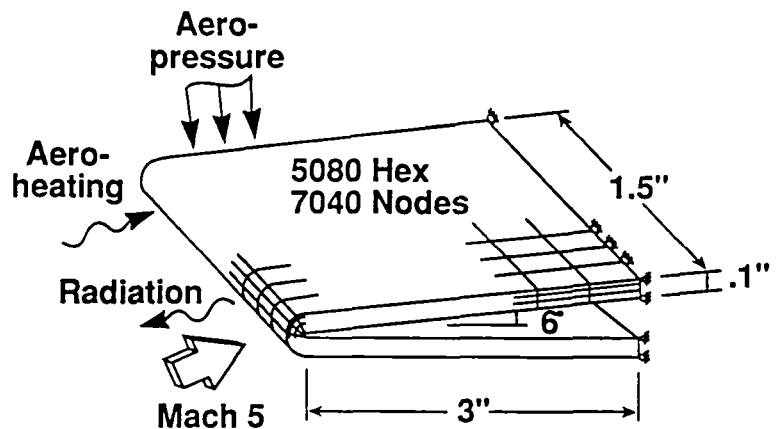
The flux-based formulation for predicting high-temperature structural response will be extended using unified viscoplasticity theory.

Publications

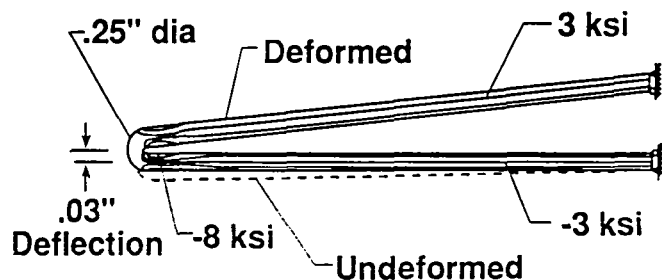
1. Pandey, A. K.; Dechaumphai, P.; and Wieting, A. R. "Thermal-structural Finite Element Analysis Using Linear Flux Formulation." AIAA Paper 89-1224-CP, Apr. 1989.
2. Dechaumphai, P.; Wieting, A. R.; and Pandey, A. K. "Fluid-Thermal-Structural Interaction of Aerodynamically Heated Leading Edges." AIAA Paper 89-1227-CP, Apr. 1989.



Relative CPU time for computing $[K]$.



Three-dimensional, leading-edge finite-element model.



Stress on deformed leading edge.

Hydrodynamics of Self-Propelled Bodies

V. C. Patel, Principal Investigator

Co-investigator: F. Stern

Iowa Institute of Hydraulic Research/University of Iowa

Research Objective

To extend the Iowa Institute of Hydraulic Research (IIHR) numerical method of solving the Reynolds-averaged Navier-Stokes equations in order to simulate the flow around complete hydrodynamic vehicles, including appendages and propulsors.

Approach

The IIHR numerical method solves the complete Reynolds-averaged Navier-Stokes equations in numerically generated, body-fitted, nonorthogonal coordinates for unsteady, three-dimensional turbulent flows. No approximations are made other than those implied by the two-equation turbulence model that is used. Several basic numerical and physical problems are being addressed (including grid generation for complex shapes and moving boundaries, time accuracy of the numerical scheme, and turbulence model development) through studies of model problems; some examples of the model problems are (1) the flow around bodies at incidence, to study the topology of three-dimensional flow attachment and separation and to evaluate turbulence models; (2) the flow around a propeller, to resolve the viscous flow over the blades as well as the flow around it; and (3) the flow over bodies with appendages and/or propellers.

Accomplishment Description

Programs developed at IIHR were first implemented on NAS facilities in late 1989. Since then, calculations have been

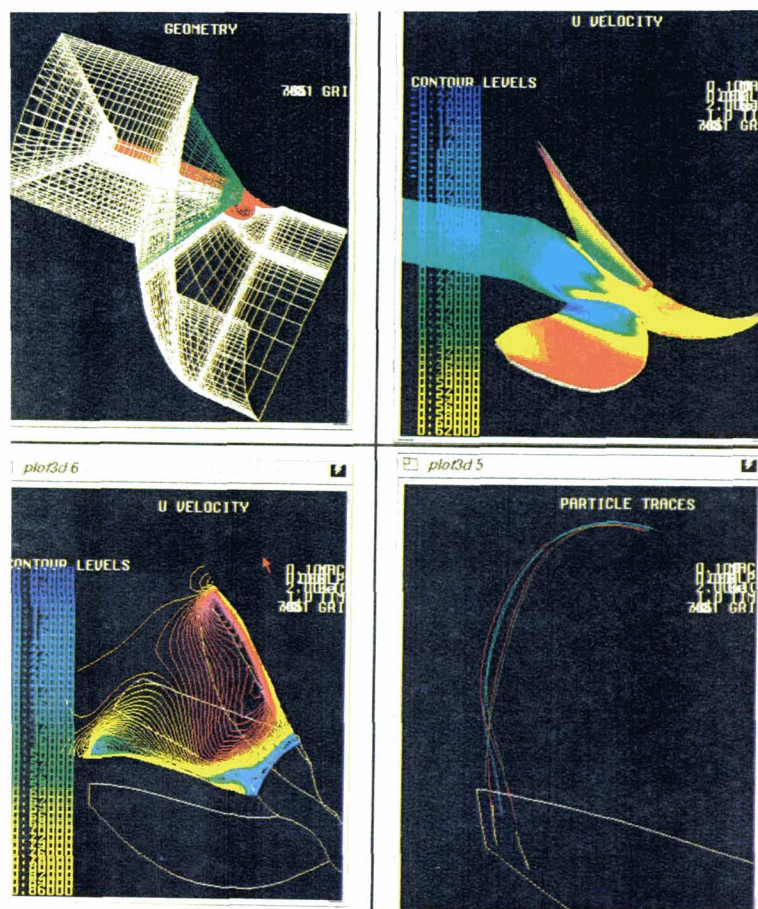
performed for (1) laminar flow on a body of revolution over a range of incidences to study the various types of flow separation that arise as the incidence is varied; (2) turbulent flow over a self-propelled appended body to study propeller/hull interaction; (3) turbulent flow for a two-dimensional cascade to evaluate the performance of a two-layer turbulence model; and (4) laminar flow over the SR-7 turboprop to study the details of the complex blade-to-blade flow.

Significance

Despite the impressive strides made in computational fluid dynamics, a numerical method that can predict the viscous flow around a complete marine vehicle does not exist, even for the relatively simple case of a submerged body in steady motion along a straight path. Neither do methods that can calculate the flow on and around a marine propeller. This research is being carried out with the sponsorship of ONR, and in support of the DARPA Subtech Program, to advance the technology base for the design of marine vehicles and propulsors.

Future Plans

We plan to continue the development of numerical methods that can calculate the flow on and around a marine propeller and predict the viscous flow around a complete marine vehicle. This research is being carried out with the sponsorship of ONR, and in support of the DARPA Subtech Program.



Navier-Stokes solution for SR-7 turboprop.

Generic National Aero-Space Plane Afterbody

James L. Pittman, Principal Investigator

Co-investigators: Kenneth E. Tatum and Mohamed E. El-Eshaky
NASA Langley Research Center

Research Objective

The ultimate goal of this study is to develop procedures for the analysis and design of generic hypersonic afterbody geometries. The prime immediate goal is to determine the optimum numerical algorithms and codes for analyzing diffusive, multiple-species, hypersonic, three-dimensional flow fields.

Approach

Several Navier-Stokes/Euler codes, using different numerical algorithms, are exercised to analyze flow about a generic scramjet nozzle/afterbody model. The calculated results, two- and three-dimensional Euler and viscous, are compared with measured surface (static) and flow-field (pitot) pressures for several single- and multiple-species flows.

Accomplishment Description

An explicit parabolized Navier-Stokes (PNS) code with multiple-gaseous-species capability was modified to allow solid-wall boundary conditions internal to the grid in order to define a cowl separating a jet flow from a free-stream flow. Calculations in the Euler mode were made to determine the accuracy obtainable with inviscid calculations. The generic scramjet nozzle/afterbody model was modeled in two and three dimensions and the computed results compared with measured pressures from the NASA Langley 20-Inch Mach 6 Wind Tunnel. The jet fluid was modeled as air (perfect gas), a mixture of nitrogen and oxygen, and two mixtures of argon and freon-12 to simulate the ratio of specific heats of actual hot

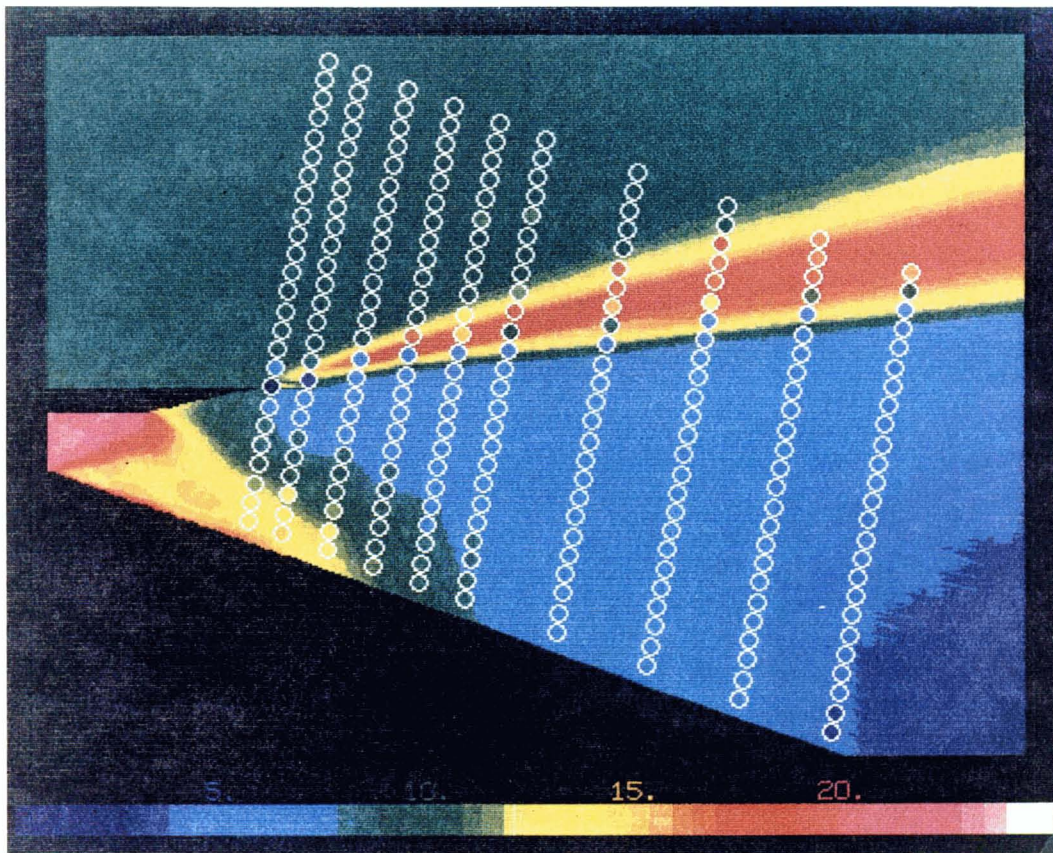
scramjet exhaust gases. The implicit GASP code was subsequently exercised in the same fashion and proved to be the more robust code for high jet pressures. The GASP 2D PNS calculations showed improved agreement with experiment over Euler solutions, both in surface pressures and flow-field details around the cowl trailing edge. A three-dimensional, explicit, marching Euler calculation on a cross-plane grid of 2100 points required 700 Cray Y-MP seconds, and a two-dimensional, implicit PNS calculation required about 9300 Cray Y-MP seconds on a 17,000-point grid. A three-dimensional, explicit, elliptic full Navier-Stokes code was modified to accept block-structured grids, and two multiple-species, half-span nozzle calculations were performed, each requiring 24.4 Cray-2 CPU hours. A total of 95 NAS hours were used during the 1989-90 operational year.

Significance

Proposed hypersonic vehicles will operate in flight regimes beyond the accessibility of ground-based test facilities. The ability to numerically analyze and design scramjet/afterbody components for such vehicles is crucial for efficient, integrated, interdisciplinary design.

Future Plans

A full three-dimensional hypersonic vehicle will be modeled, including a precisely defined scramjet nozzle and afterbody. Parametric studies will be conducted on families of two-dimensional afterbody shapes. The GASP code will be coupled with an iterative optimization/design code.



Euler scramjet exhaust calculation compared with measured flow-field pitot pressures; argon/freon-12 simulant gas jet ejecting into a Mach 6 air free stream.

ORIGINAL PAGE
COLOR PHOTOGRAPH

NASP-TMP Forebody/Inlet Integration

James L. Pittman, Principal Investigator
Co-investigator: Lawrence D. Huebner
NASA Langley Research Center

Research Objective

To computationally analyze the flow field for hypersonic forebody/inlet/afterbody regions as part of the overall National Aero-Space Plane (NASP) technology maturation plan (TMP) effort in the area of propulsion/airframe integration (PAI).

Approach

An Euler/Navier-Stokes computational fluid dynamics (CFD) code is used to compute the three-dimensional flow about a hypersonic forebody-alone, a full body, a forebody with blocked inlet, and a forebody with inlet fairing, and the two-dimensional flow about a "powered" hypersonic vehicle. The CFD solutions are compared with surface and pitot pressure data, as well as with Schlieren and oil-flow photographs.

Accomplishment Description

Pitot pressure measurements made at the inlet plane of the forebody-alone showed very good agreement with pitot pressures calculated from parabolized Navier-Stokes (PNS) solutions. At four different conditions the computational results accurately predicted boundary-layer thickness and the extent of the classic boundary-layer accumulation on the centerline. Each of these solutions took about 12 megawords of memory and 40 Cray-2 minutes of CPU time. The PNS solutions for a NASP-type full body at Mach 20 in helium also showed good agreement of centerline surface pressures. These solutions took about 16 megawords and one Cray-2 hour. Thin-layer Navier-Stokes (TLNS) solutions of the forebody with blocked inlet and with faired inlet were performed with expectations of predicting massive flow separation on the forebody under-surface. Comparison of computational surface particle traces with oil-flow photographs showed excellent agreement of the extent of separation, flow patterns with the separated region, and the location of flow reattachment on the blocked- and faired-inlet surfaces. These solutions required 21 megawords and the CPU time varied from 5 to 18 Cray-2 hours. Finally, a two-dimensional study was undertaken to analyze flow-field features associated with a simulated powered hypersonic

vehicle. Euler and PNS solutions were performed for four different test conditions. Under one condition, an additional shock was seen underneath the cowl trailing edge and was anticipated to be from boundary-layer separation caused by localized recirculation. An effort was made to compute this using the code as a TLNS solver, and as shown in the figure, accurate prediction of the salient flow-field features was achieved. All two-dimensional solutions took about 8 megawords of memory. The Euler solution required about 5 Cray Y-MP minutes, the PNS solutions required about 52 Cray Y-MP minutes, and the TLNS solution required 18 Cray Y-MP hours.

Significance

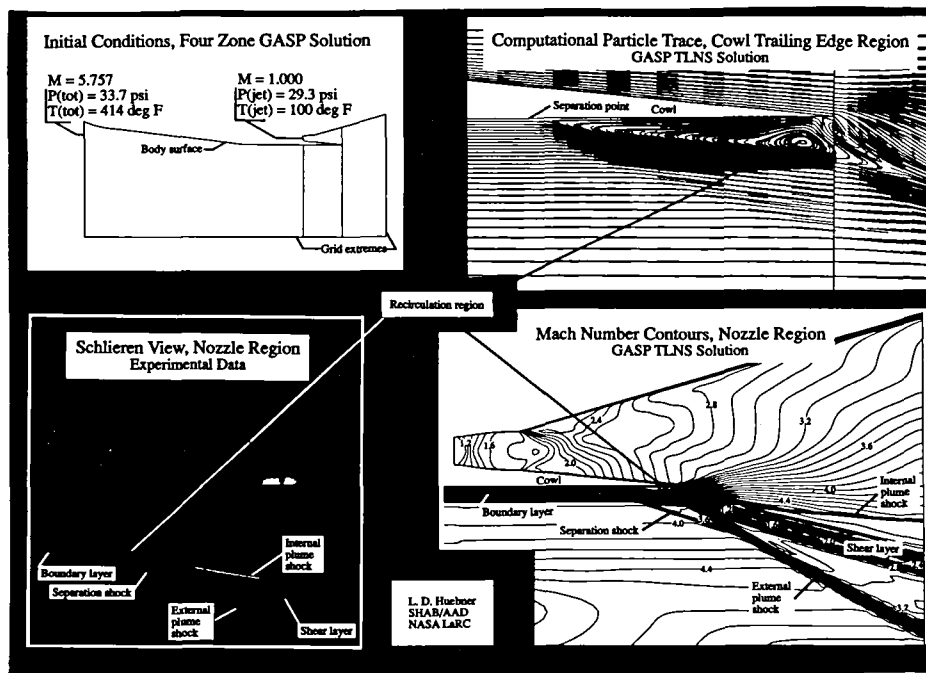
The ability of computational methods to accurately predict flow phenomena associated with NASP PAI is critical to the design of the NASP vehicle. The accurate comparison of CFD results with wind tunnel data provides confidence and the first step in the code's ability to predict the flow about NASP-type vehicles at full-scale flight conditions.

Future Plans

Computations on the simulated powered configuration will be expanded to three dimensions and will be executed with a turbulence model. This effort will eventually result in tip-to-tail, three-dimensional Navier-Stokes solutions on the entire NASP configuration.

Publications

1. Huebner, Lawrence D. "Computational Investigation of LaRC Test Technique Demonstrator Reentry Configurations (U)." Paper 36, 7th NASP Technology Symposium, Cleveland, OH, Oct. 1989.
2. Woods, William C.; Huebner, Lawrence D.; and Everhart, Joel L. "Measured and Predicted Pressure Distributions on the Langley Test Technique Demonstrator at M=20 in Helium (U)." Paper 33, 7th NASP Technology Symposium, Cleveland, OH, Oct. 1989.



Nozzle flow-field prediction for a hypersonic air-breathing vehicle.

Development and Evaluation of Computational Methods for Turbomachinery Applications

Richard H. Pletcher, Principal Investigator

Co-investigator: Edward J. Hall

Iowa State University/Allison Gas Turbine Division, General Motors Corporation

Research Objective

To develop and evaluate advanced numerical methods for the simulation of unsteady viscous flows occurring in turbomachinery applications.

Approach

Three traditional central-difference algorithms and a new upwind-biased, generalized conjugate gradient, total-variation-diminishing (GCG-TVD) scheme have been used to solve the two-dimensional, time-dependent Navier-Stokes equations for steady and unsteady flows.

Accomplishment Description

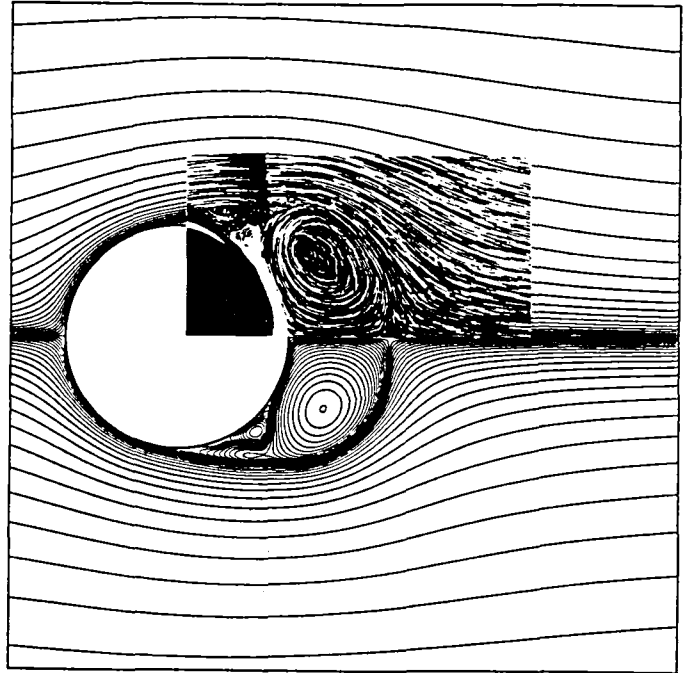
Unsteady Navier-Stokes flow simulations were carried out on a fine (361×99) grid for an impulsively started cylinder at Reynolds numbers of 3000 and 9500. The results from four algorithms were compared with experimental data. The purpose was to compare the accuracy and efficiency of the schemes for complex time-dependent flow. Three of the algorithms used central differences, the explicit hopscotch and Runge-Kutta schemes, and an approximately factored, linearized block implicit scheme. The fourth scheme was a new upwind-biased algorithm utilizing an implicit time-marching relaxation procedure based on Newton iteration. This upwind scheme utilizes a TVD formulation and a GCG minimal residual descent matrix relaxation procedure. The accompanying figure compares the streamline pattern predicted by the upwind scheme with flow visualization results after the cylinder had moved two diameters from rest at a Reynolds number of 9500. Such a calculation requires about one hour of Cray-2 time and 4 megawords of memory.

Significance

Current turbomachinery design analyses are generally based on steady flow calculations although most real turbomachinery flows are unsteady. With recent advances in computer architecture it is becoming feasible to simulate increasingly complex flows. The capabilities and limitations of algorithms for time-dependent calculations need to be understood so that time-accurate calculations can ultimately be used to guide the design of more efficient turbomachines.

Future Plans

Plans include the development and evaluation of time-accurate algorithms for three-dimensional flows, along with some improvements in the efficiency of the upwind scheme.



A GCG-TVD-predicted streamline pattern for the flow of an impulsively started cylinder; $Re = 9500$, $t = 2.0$.

Vapor-Phase Growth of Crystals in Microgravity

Andrew Pohorille, Principal Investigator
University of California, Berkeley

Research Objective

The objective of this research is to explore the factors that influence the structure of thin films grown by molecular deposition from the vapor phase. The ultimate goal is to explain the results of experiments conducted in space by the 3M Corporation, which showed that thin films of copper phthalocyanine formed in microgravity are different from those formed in the presence of gravity.

Approach

Deposition of particles on the substrate and their diffusion on the surface was simulated by a molecular dynamics method. New particles were introduced to the system in the direction perpendicular to the deposition surface and moving toward this surface. The velocities of the molecules at the deposition surface were reset periodically to a Maxwellian distribution corresponding to the desired temperature of the substrate.

Accomplishment Description

(1) Deposition of the spherical particles with the Lennard-Jones potential of interactions led to well-layered crystalline growth. This regular growth was perturbed only when new particles were introduced to the systems in very short succession. In those conditions particles reaching the substrate did

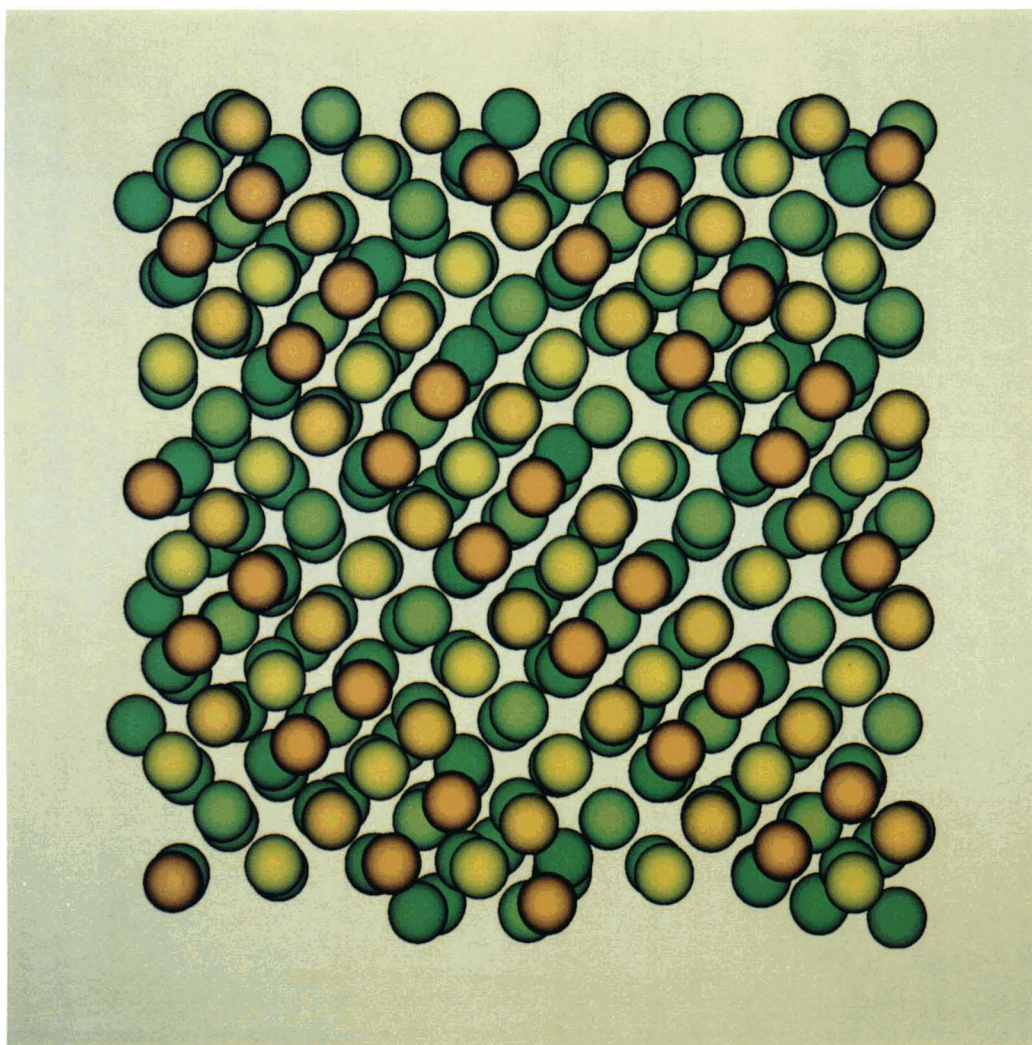
not have time to relax on the surface, and disordered, but uniform, growth was observed. (40 CPU hours, 4 megawords memory) (2) Six-membered ring molecules interacting via Lennard-Jones potentials tended to form regular crystal-like structures if sufficient time was permitted for surface diffusion. (50 CPU hours, 8 megawords memory) (3) Spherical particles introduced with nonuniform spacial distributions and non-Maxwellian velocity distributions also exhibited crystalline growth if the deposition rate was low. (60 CPU hours, 4 megawords memory)

Significance

Our studies on simple model systems showed that (1) molecular dynamics is a very good tool for detailed studies of vapor-phase deposition, (2) the main factor influencing thin film growth is the relaxation time on the surface, and (3) changes in the deposition parameters that can be attributed to convective flow appear not to change the structure of thin films.

Future Plans

We will apply molecular dynamics to simulate deposition and diffusion on the surface of copper phthalocyanine using recently developed, realistic intermolecular potential functions.



Atoms deposited on the surface from the vapor phase form crystalline-like structure. Colors of atoms change from orange (top) to green (bottom) with their depth in the sample. For better visibility, atomic radii were reduced by 50%.

Finite-Element Analysis of Equilibrium and Nonequilibrium Flows

Ramadas K. Prabhu, Principal Investigator

NASA Langley Research Center/Lockheed Engineering and Sciences Company

Research Objective

To develop a capability to predict aerothermal loads experienced by bodies in high-speed flight.

Approach

Navier-Stokes equations in two and three dimensions will be solved by the finite-element method on meshes generated with an adaptive remeshing technique. The gas properties will be modeled with the assumptions of ideal gas as well as those of thermal and chemical equilibrium and nonequilibrium.

Accomplishment Description

A two-dimensional finite-element Navier-Stokes solver developed for ideal gases was modified to include equilibrium flows. The adaptive remeshing procedure proved to be an excellent method for capturing sharp flow features. This code was applied successfully to solve several hypersonic flows involving shock interference at Mach 8 and Mach 16 past a circular cylinder. (The cylinder is the typical shape of a leading edge of a winglike body.) Results compared well with

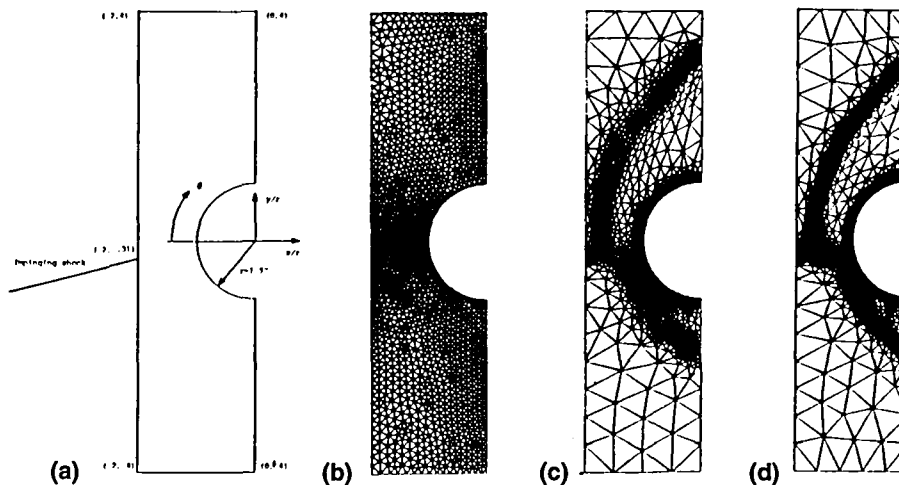
available experimental data. Depending on the size of the mesh, it took about 5 megawords of memory and 2 to 3 hours of computing time for convergence on each mesh. Extension of this code to chemical nonequilibrium flows is in progress.

Significance

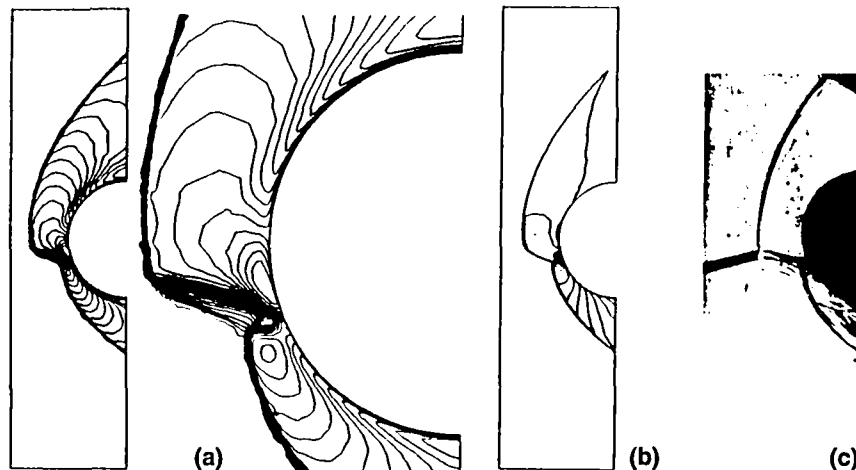
Flight conditions of hypersonic vehicles are such that the classical ideal-gas assumption is not valid. During extreme cases, thermal and chemical nonequilibrium effects play a significant role. Under these conditions the vehicles experience high aerodynamic and thermal loads. Knowledge of these loads and design options to alleviate them is necessary for a successful design.

Future Plans

The existing computer codes will be extended to include thermal and chemical nonequilibrium flows and effects of turbulence. Several problems of interest, including the cooling of a cylindrical leading edge by blowing into the stream, will be studied.



Computational domain and finite-element meshes used for a Type IV shock interference problem on a cylinder: (a) computational domain, (b) initial mesh, (c) adaptively generated second mesh, (d) adaptively generated third mesh—7614 nodes, 4224 triangles, 6449 quads.



Results for a Type IV shock interference problem; $M = 16$, $Re = 1.45 \times 10^5$: (a) velocity contours, (b) pressure contours, (c) Schlieren photograph.

Chaotic, Unsteady, Low-Reynolds Number Navier-Stokes Flow

Thomas H. Pulliam, Principal Investigator
NASA Ames Research Center

Research Objective

To investigate unsteady computations of low-Reynolds number flow past a realistic (two-dimensional airfoil) geometry, and to study the physics of the flow using nonlinear dynamics.

Approach

The two-dimensional full Navier-Stokes equations were solved using either explicit or implicit time differencing and second-, fourth-, or sixth-order-accurate space differences. Long-term time integrations were performed in which hundreds of thousands of time steps were obtained for analysis.

Accomplishment Description

Standard nonlinear dynamics analysis tools were applied to the unsteady characteristics of numerically computed low-Reynolds number flow past a two-dimensional airfoil. Numerical results were generated using ARC2D and other Navier-Stokes codes. Results for a NACA 0012 airfoil at $M_\infty = 0.2$, $\alpha = 20^\circ$, and $Re = 800$ to 3000 exhibit a progression from

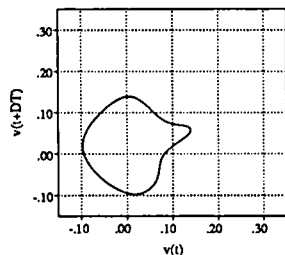
single-period vortex shedding to chaotic unsteadiness via a sequence of period doublings. Between $Re = 1600$ and $Re \approx 2200$ periodic windows are observed, with subsequent doublings back to chaos. Time-delay reconstructions, Poincaré sections, frequency decompositions, Lyapunov exponent calculations, and dimension calculations are used to characterize the flow.

Future Plans

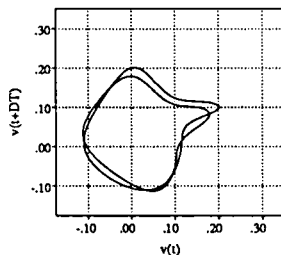
A more detailed analysis of the mechanisms associated with the bifurcation sequence to chaos will be carried out. Local Lyapunov analysis will be used to characterize the critical physical events responsible for the cascade.

Publications

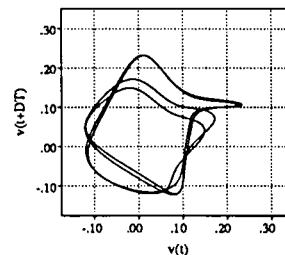
Pulliam, T. H., and Vastano, J. A. "Numerical Simulation of Chaotic Flows: Measures of Chaos." Presented at the 3rd Joint ASCE/ASME Mechanics Conference Forum on Chaotic Flows, La Jolla, CA, 1989.



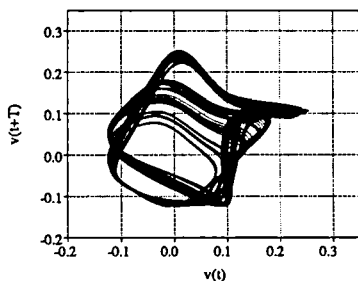
$Re = 800$, Period 1



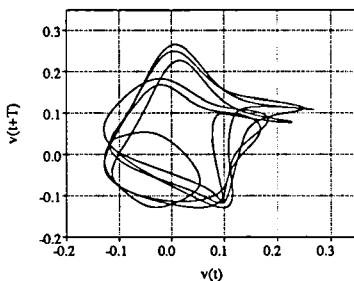
$Re = 1075$, Period 2



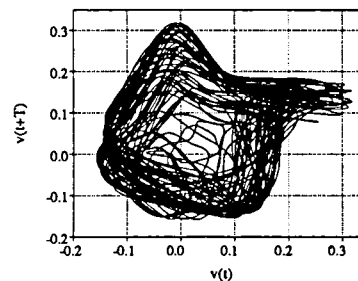
$Re = 1400$, Period 4



$Re = 1600$, Low D Chaotic



$Re = 1825$, Period 6



$Re = 3000$, Chaotic

Time-delay reconstructions of the vertical velocity near the trailing edge of a NACA-0012 airfoil, showing the period-doubling sequence to chaos; $M_\infty = 0.2$, $\alpha = 20^\circ$.

A Finite-Difference Approach to Direct Simulations and Large-Eddy Simulations of Turbulent Flow

Man Mohan Rai, Principal Investigator
NASA Ames Research Center

Research Objective

To develop finite-difference methods that can be used for direct and large-eddy simulations of compressible turbulent flow.

Approach

The principle obstacle to using existing finite-difference schemes for direct and large-eddy simulations of turbulent flow is the inadequate accuracy levels of such schemes. In this study the use of high-order-accurate finite-difference methods is investigated.

Accomplishment Description

A fifth-order-accurate upwind finite-difference method was developed for direct and large-eddy simulations of compressible, turbulent flow. The truncation error of this scheme is sufficiently small to yield accurate simulations of turbulent flow with a reasonable number of grid points. A variant of this method that is applicable to incompressible flows was used to simulate low-Reynolds number, fully developed, turbulent channel flow. The results thus obtained compare well with the results of earlier simulations using the spectral method. The version of the method for compressible flows was used to develop a code to compute transitional/turbulent flow over a flat plate. Preliminary results were obtained on a coarse grid for a free-stream Mach number of 0.1 and Reynolds number of 50,000.0/in. The figure shows the skin friction distribution on the plate as a function of the distance along the plate. The computed data appears "noisy" because it has not been averaged in time. The transition region compares well with the experimental data. The skin friction values in the turbulent region are a little lower than the data. Grid refinement studies indicate that a computation on a finer grid will yield the right distribution of skin friction. A fine-grid computation will require about 15 million grid points (which translates into 16 megawords of run-time memory) and about 3000 hours of Cray-2 single-processor hours.

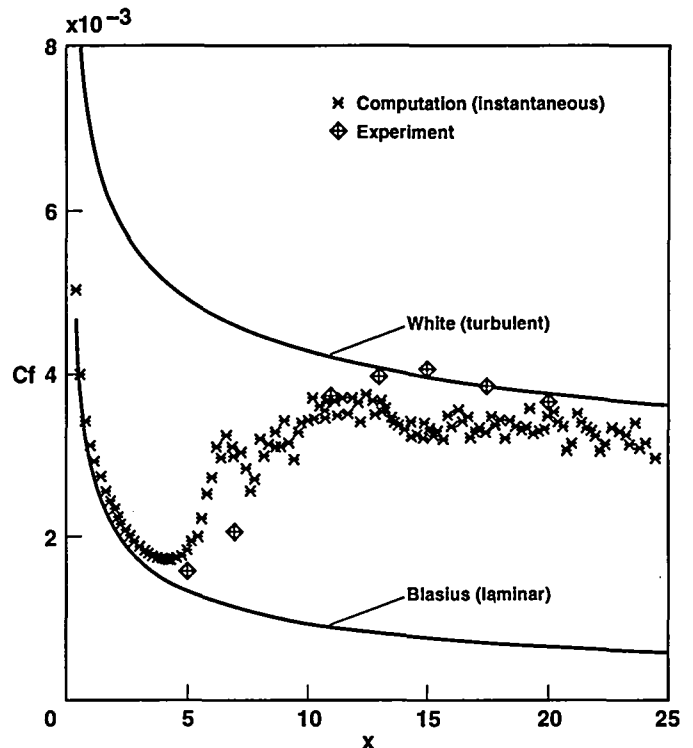
Significance

To date the most successful direct simulations have been performed with spectral methods because of the very high accuracies that these methods possess. However, spectral methods are difficult to use for complex geometries. The high-

order-accurate finite-difference method developed in this study can, with modifications, be used in a straightforward manner for complex geometries. In addition, the direct simulation of transition on a flat plate being attempted in this study represents a first-of-a-kind effort in computational fluid dynamics.

Future Plans

The flat-plate computation will be performed on a more refined mesh.



Skin friction distribution on a flat plate; $M = 0.1$, $Re = 50,000/\text{in.}$, number of grid points = 10,875,385.

Three-Dimensional Multi-Airfoil Navier-Stokes Simulations of Rotor/Stator Interaction in Turbines

Man Mohan Rai, Principal Investigator

Co-investigators: Nateri K. Madavan and Akil A. Rangwalla
NASA Ames Research Center

Research Objective

To develop and validate a three-dimensional Navier-Stokes code (ROTOR-4) to simulate flows through single-stage turbomachinery with an arbitrary number of blades or vanes in each row.

Approach

The ROTOR-4 code incorporates multiple-zone technology and an implicit, high-order-accurate upwind scheme for the time-accurate solution of the thin-layer Navier-Stokes equations.

Accomplishment Description

The development of the ROTOR-4 code was completed and validation studies for an axial turbine configuration with three stator and four rotor airfoils are in their final stage. Mesh refinement studies were also carried out. The stator:rotor airfoil count (3:4, which is equivalent to 21:28) in the calculation closely approximates that of the experimental configuration (22:28). The accompanying photograph shows the instantaneous static pressure distribution in this turbine stage from

inlet (left) to exit (right). The multi-airfoil calculation requires approximately 150 Cray Y-MP single-processor hours; it requires under 4 megawords of core memory and 27 megawords of auxiliary memory (SSD).

Significance

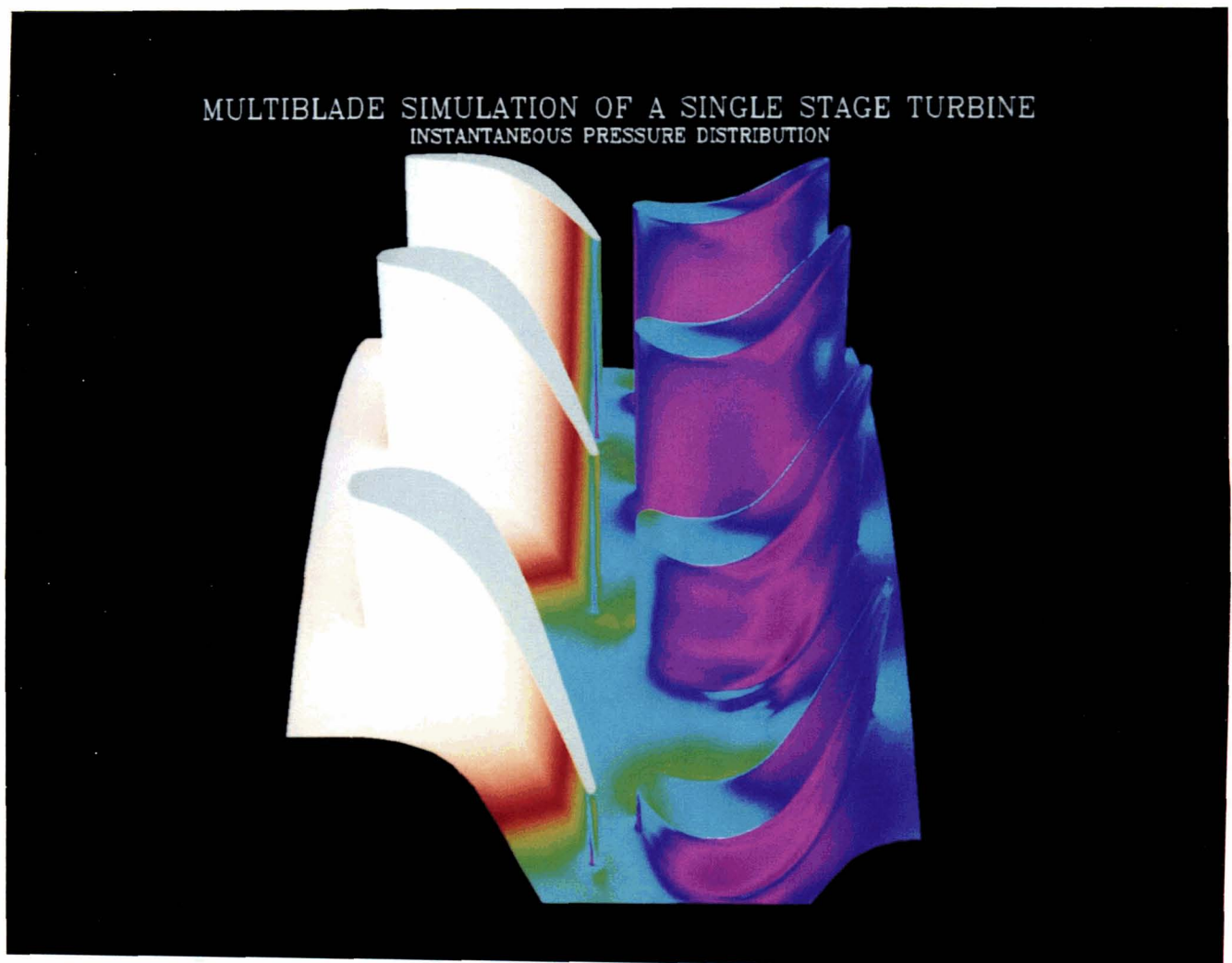
The ROTOR-4 code is the only code of its kind and is of extreme importance to the turbomachinery community in understanding the unsteady aerodynamics of turbomachinery. The validation studies conducted represent the most comprehensive numerical simulation to date of a single-stage turbine.

Future Plans

Efforts are under way to develop a multistage, multi-airfoil code (STAGE-3) using ROTOR-4 as the base code.

Publications

Madavan, N. K.; Kelaita, P.; and Gavali, S. "Supercomputer Applications in Gas Turbine Flowfield Simulation." To be published in *The International Journal of Supercomputer Applications* 4.2 (1990).



Multiblade simulation of a single-stage turbine; instantaneous pressure distribution.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Unsteady Flow in a Multistage Compressor

Man Mohan Rai, Principal Investigator

Co-investigators: Karen L. Gundy-Burlet and Akil Rangwalla

NASA Ames Research Center

Research Objective

To develop a two-dimensional Navier-Stokes zonal code (STAGE-2) to simulate the unsteady flow within multistage turbomachines.

Approach

The unsteady, thin-layer Navier-Stokes equations are solved using an implicit, third-order-accurate, upwind-biased finite-difference scheme and a multiple-zone methodology.

Accomplishment Description

A two-dimensional multistage turbomachinery code (STAGE-2) was developed and validated. Results from the code compare well with experimental data for both a multistage compressor and a single-stage turbine. Several calculations, including grid and time-stop refinement studies, were run on the Cray Y-MP and the Cray-2. The results of these investigations can be found in publications 1 and 2. The accompanying photograph shows instantaneous entropy contours within a 2 1/2-stage compressor. Features of interest include wake/wake and wake/airfoil interactions. A typical calculation requires 2 megawords of run-time memory, 5 megawords of SSD, and between 5 and 40 hours of Cray-2 single-processor hours, depending on the level of convergence required.

Significance

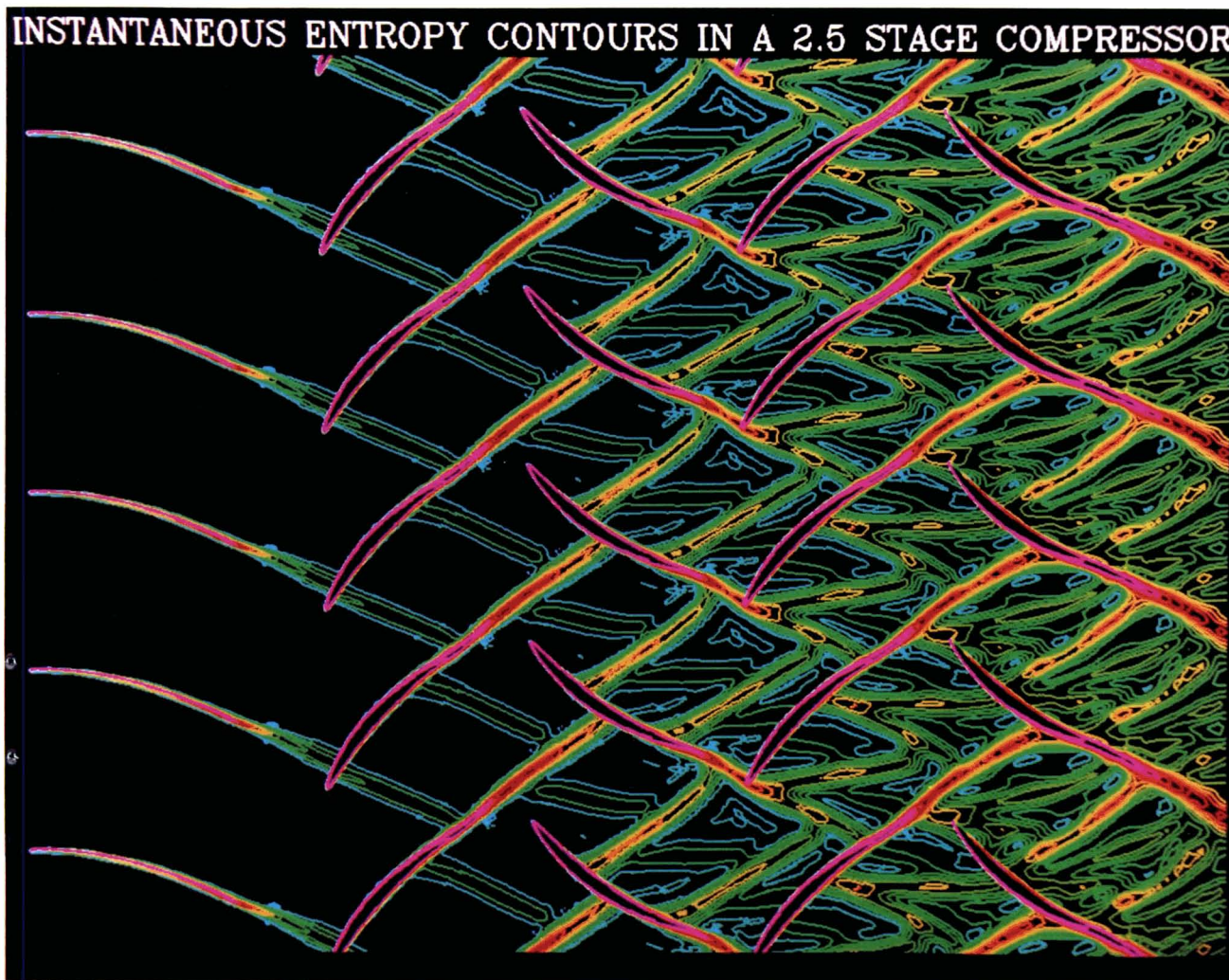
The development of STAGE-2 represents a major step in computing flows through multistage turbomachines. As seen in the photograph, the flow increases in complexity as it moves downstream through the compressor. Therefore, in order to simulate the flow in a downstream stage using a single-stage or cascade code, complex boundary conditions would have to be applied at the stage inlet. Hence, multistage turbomachinery codes are required in order to understand the aerodynamic processes in multistage turbomachines.

Future Plans

A three-dimensional multistage turbomachinery code, STAGE-3, will be developed and validated during the 1990-91 NAS operational year.

Publications

1. Gundy-Burlet, K. L.; Rai, M. M.; and Dring, R. P. "Two-Dimensional Computations of Multi-Stage Compressor Flows Using a Zonal Approach." AIAA Paper 89-2452, July 1989.
2. Gundy-Burlet, K. L.; Rai, M. M.; Stauter, R. C.; and Dring, R. P. "Temporally and Spatially Resolved Flow in a Two-Stage Axial Compressor, Part 2 - Computational Assessment." ASME Paper 90-GT-299, June 1990.



Instantaneous entropy contours in a 2 1/2-stage compressor.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Analysis of Rotary Engine Processes and Computer Codes

M. S. Raju, Principal Investigator
Sverdrup Technology, Inc.

Research Objective

The direct-injection stratified-charge rotary combustion engine (DISC-RCE) would be a desirable powerplant for light aircraft and drones (including high-altitude applications) if its efficiency could be made closer to that of reciprocating engines. Continuing research and development sponsored by NASA is aimed at reducing the cruise brake-specific fuel consumption from the current value of 0.42 lb/bhp-hp to 0.35 or less by the end of 1992.

Approach

Much of the expected improvement in RCE performance will be facilitated by computational fluid dynamics-driven fuel-injection, spray, and nozzle optimization; rotor pocket and nozzle relocations; and related modifications.

Accomplishment Description

A new computer code was developed for predicting the turbulent and chemically reacting flows occurring inside a DISC-RCE. The solution procedure is based on an Eulerian-Lagrangian approach in which the unsteady, three-dimensional Navier-Stokes equations for a perfect-gas mixture with variable properties are solved in generalized, Eulerian coordinates on a moving grid by making use of an implicit, finite-volume, Steger-Warming flux-vector-splitting scheme, and the liquid-phase equations are solved in Lagrangian

coordinates. Numerical predictions of the pressure and temperature histories, the torque generated by non-uniform pressure distribution within the chamber, energy release rates, and various flow-related phenomena were obtained and compared with experimental data. The code takes about 10 CPU hours and 5 megawords of memory when the calculations are performed on a grid with a mesh size of $31 \times 16 \times 20$ on a Cray Y-MP for one entire cycle.

Future Plans

The computer code will be modified with the incorporation of a k- ϵ turbulence model and an eddy break-up model to account for some of the effect of turbulence on the combustion model. The spray model may also be improved with the inclusion of submodels associated with the droplet collision, droplet break-up, and droplet dispersion resulting from turbulence.

Publications

1. Raju, M. S., and Willis, E. A. "Analysis of Rotary Engine Combustion Processes Based on Unsteady, Three-Dimensional Computations." AIAA Paper 90-0643, 1989.
2. Raju, M. S., and Willis, E. A. "Computational Experience with a 3-D Rotary Engine Combustion Model." *Proceedings of the 1990 Joint AIAA/FAA Symposium on General Aviation Systems*. Ocean City, NJ, Apr. 1990.



Analysis of rotary combustion engine processes.

ORIGINAL PAGE
COLOR PHOTOGRAPH

An Adaptation Procedure Combining Mesh Refinement with Mesh Movement for Compressible Flows

R. Ramakrishnan, Principal Investigator
NASA Langley Research Center

Research Objective

To use a mesh adaptation scheme that combines mesh refinement and derefinement with mesh movement to capture sharp gradients such as shocks by using an upwind finite-element formulation.

Approach

A mesh adaptation scheme that uses both triangular and quadrilateral elements to enrich and coarsen elements based on refinement indicators has found an application in modeling inviscid and viscous flows. To minimize the number of elements used in the analysis, this adaptive mesh-refinement procedure is complimented by a mesh movement scheme. Mesh movement is accomplished by using the concept of a spring-mass system centered on each node. Stiffness coefficients for these springs are related to element errors, which are computed from normalized second derivatives of a key variable such as density. To demonstrate the adaptation scheme, Mach 6 inviscid flow over a cylindrical leading edge is modeled. A node-based, upwind finite-element algorithm with higher order extensions is used to solve the compressible Euler equations. The finite-element mesh obtained after two levels of refinement contains 3769 nodes, and the density contours on this mesh appear on the left side of the figure. The mesh obtained after one refinement with mesh movements at specified intervals contains 1587 nodes. This mesh and the density contours on it appear on the right side of the figure.

The surface pressure distributions along the right boundary of the computational domain compare well for the two cases. The inclusion of mesh movement produces a crisper definition of the bow shock. A closer look at the elements in the vicinity of the bow shock shows the mesh being aligned and stretched along the shock when adaptive refinement is combined with mesh movement.

Accomplishment Description

An adaptation procedure that combines mesh refinement with mesh movement is coupled with an upwind, higher order finite-element algorithm to model high-speed compressible flow.

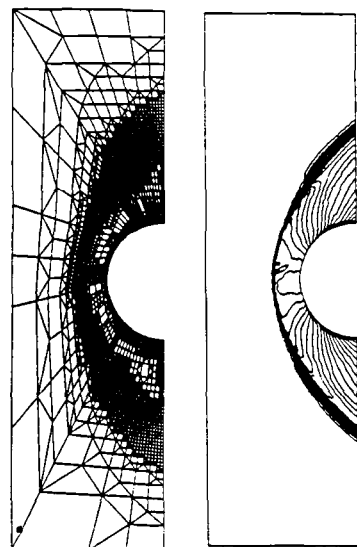
Significance

The mesh refinement procedure augmented by nodal relocation orients the elements along the shocks or flow discontinuities. Nodal density across such high gradients is increased, resulting in excellent shock resolution. The number of elements needed to model the domain is reduced substantially and no a priori knowledge of the flow field is required.

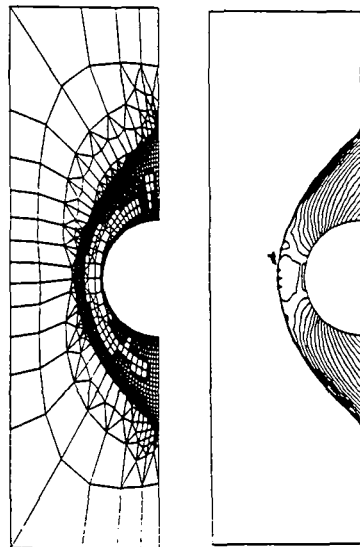
Future Plans

Future plans include developing accurate indicators for mesh movements in the boundary layer for viscous flows and extending the adaptation procedure for three-dimensional applications.

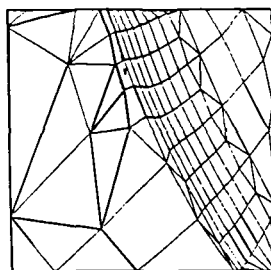
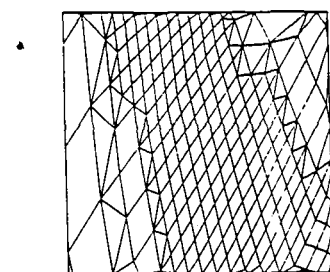
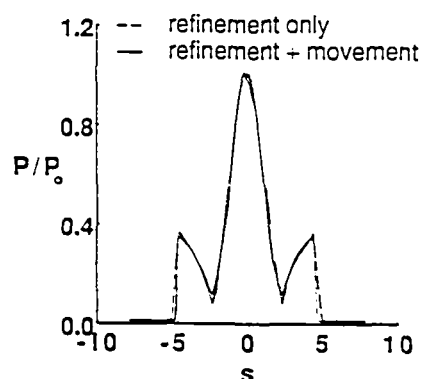
Refinement only.



Surface pressure distributions.



Refinement plus movement.



Mesh refinement for inviscid flow over a cylindrical leading edge.

Modeling Global Cloudiness

David A. Randall, Principal Investigator
Colorado State University

Research Objective

To numerically simulate the global distribution of cloudiness in the Earth's atmosphere and the effects of the clouds on climate—specifically, the effects of the clouds on CO₂-induced climate change.

Approach

A three-dimensional atmospheric general circulation model is used in low-resolution simulations of several annual cycles and in high-resolution simulations of individual seasons.

Accomplishment Description

A five-year low-resolution simulation was completed, and the model is now being run with increased resolution, using both Navier-Stokes and Reynolds codes. Runs require 12 megawords of memory and about 36 CPU hours per simulated year. This year we established, among other things, that the

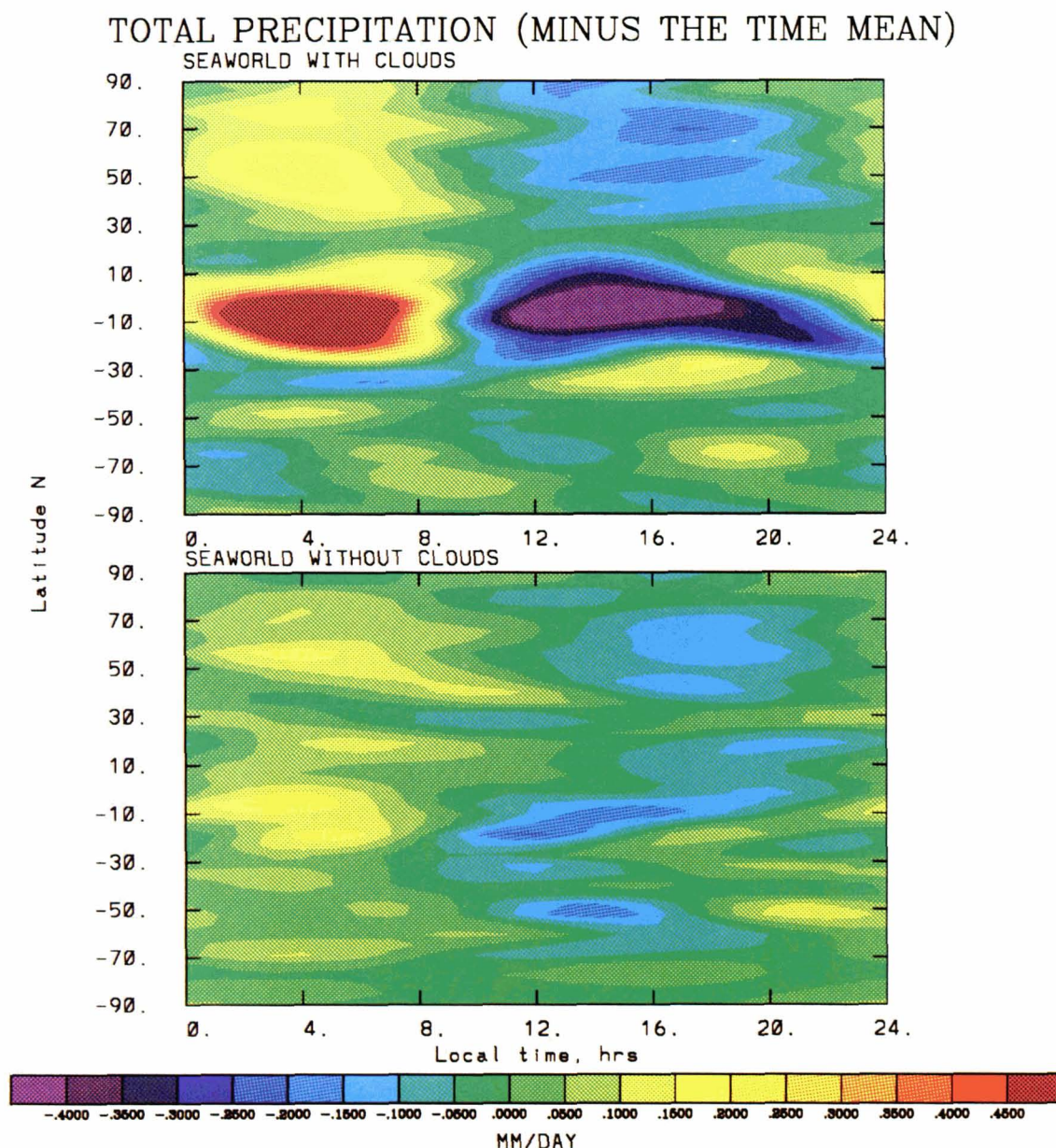
observed diurnal variation of precipitation over the oceans is due primarily to the radiative effects of clouds. This is illustrated in the accompanying figure. We have continued our participation in a comparison of results from 18 climate models from around the world.

Significance

Climate change resulting from increasing atmospheric carbon dioxide cannot be understood or predicted without taking into account the effects of the clouds.

Future Plans

The high-resolution model will be run through 18 simulated months, about 6 months of which have already been completed. Results will be analyzed with special attention to the simulated cloudiness and surface energy budget.



Total precipitation (minus the time mean).

Computation of ASTOVL Flow Fields

James A. Rhodes, Principal Investigator
McDonnell Aircraft Company

Research Objective

The objective of this research is to compute flow fields related to the design of Advanced Short Takeoff and Vertical Landing (ASTOVL) aircraft. In particular, the computation of three-dimensional jets exiting from a wall into a low subsonic crossflow and the computation of three-dimensional impinging jets are of high interest. Our 1989 objective was to apply the MCAIR upwind scheme NASTD to the above two problems and evaluate the results.

Approach

The approach in 1989 was first to attack the problem of a three-dimensional jet in a subsonic crossflow and then to proceed to the more complicated problem of the three-dimensional impinging jet. The first problem is one of the simplest jet-in-crossflow problems and yet is rich in interesting fluid dynamics. A two-zone grid was developed, which we believed to be adequate for resolving the key flow-field features. The flow solver was applied and run in a sequenced mode on a coarse version of the grid. A preliminary solution was obtained and examined.

Accomplishment Description

A grid was developed with a reflection boundary aligned parallel to the crossflow and normal to the wall. The

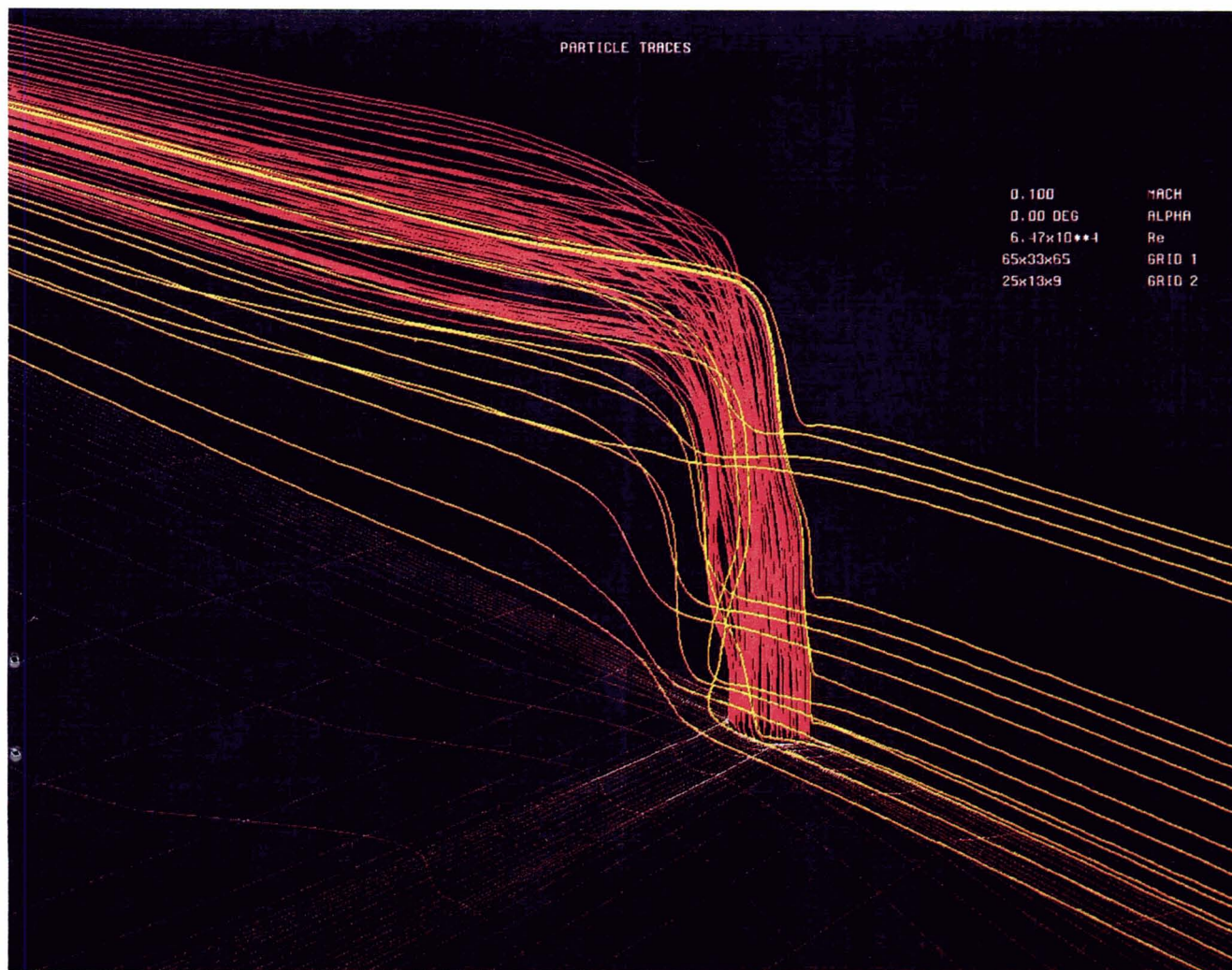
$65 \times 33 \times 65$ rectangular zone 1 is where the jet interacts with the crossflow, and the semicircular $25 \times 13 \times 9$ "nozzle" zone 2 sets up the jet and is coupled using a point-matching interface. The grid was developed using elliptic methods so that the grid in the hole is a Cartesian subset of the wall boundary. The flow solver was run on a coarse grid to obtain an initial solution, which is shown in the figure. This solution will be refined in the future.

Significance

A solution of a jet exiting from a wall into a low subsonic crossflow was obtained using a code that solves the full Reynolds-averaged Navier-Stokes equations using a finite-volume upwind scheme with an algebraic turbulence model. The solution shows many of the physical features that are characteristic of such flows.

Future Plans

This work will continue in 1990. Grid modifications will be attempted, to reduce the boundary layer thickness of the jet as it exits from the nozzle zone below the wall. Adaptive gridding will be used and the flow solver will be run using a two-equation low-Reynolds number turbulence model. We will also compute the flow inside the lift nozzle return "cactus" duct of an ASTOVL vehicle.



A three-dimensional jet in a subsonic crossflow.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Kinetic Theory Model for the Flow of a Simple Gas from a Three-Dimensional Axisymmetric Nozzle

Ben R. Riley, Principal Investigator
University of Evansville

Research Objective

The objective is to develop a model to predict the flow field of exhaust plumes for small low-thrust nozzles. In particular, the effect of the assumptions about the boundary layer along the inside wall, especially near the nozzle lip, on the flow fields external to the nozzle is of interest.

Approach

The mathematical framework for the description of the gas flow is the time-independent Boltzmann equation for binary collision. The complete scattering term is replaced by a Krook-type (collision relaxation) approximation to simplify the collision calculations. Starting with a zeroth-iteration set of flow-field parameters for temperature, number density, and mean velocity at each grid point, the method of characteristics is used to numerically solve the Boltzmann equation in order to find the first iteration values for temperature, number density, and mean velocity. This iteration procedure is repeated until a reasonable convergence is obtained. A major concern in this technique is the proper number and location of grid points, especially near the nozzle lip. The accompanying figure shows the number density in a half-plane through the centerline. Each solution required about 10 hours of Cray Y-MP time and 2 megawords of memory.

Significance

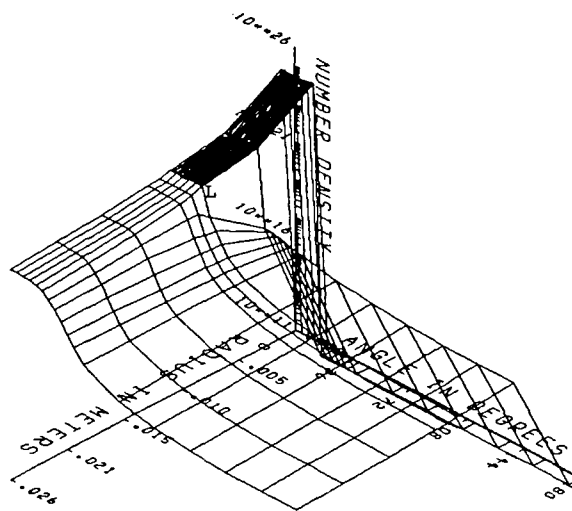
Self-induced molecular contamination around a spacecraft can limit its usefulness, because the contaminants may adversely affect components or experiments being performed on or near the spacecraft. Since it is difficult to perform ground-based experiments because of low-vacuum requirements, computer models are needed to help design the nozzles.

Future Plans

The code will be used to study the plume flow fields with different nozzle conditions. At the same time, work is being done on the code to further vectorize it and to decrease the run time.

Publications

Riley, Ben. "Kinetic Theory Model for the Flow of a Simple Gas from a Three-Dimensional Axisymmetric Nozzle." Presented at the 17th International Conference for Rarefied Gas Dynamics, Aachen, Germany, July 1990.



Three-dimensional plot of number density. The number-density axis is marked in powers of 10, and the shaded area is inside the 20°-half-angle nozzle. The angle is measured in degrees from the centerline, the nozzle-lip grid point is marked L, and the distance from the nozzle throat to the exit plane is ~0.01 m.

Scattering from Ocean Surfaces and Near-Surface Objects

Charles L. Rino, Principal Investigator

Co-investigators: Hoc D. Ngo and Thomas L. Crystal
Vista Research, Inc.

Research Objective

To develop the capability to simulate electromagnetic and acoustic scattering from dynamically evolving nonlinear ocean surfaces and near-surface objects.

Approach

Linear ocean-surface realizations are generated by allowing noninteracting Fourier modes to propagate according to the linear dispersion relation for surface waves. The linear surfaces are then converted to their nonlinear form by using an efficient procedure based on Hilbert transforms. Scatter is simulated by solving the surface boundary-value problem numerically. Scatter from a nearby object is accommodated by a new technique, which we call the mutual interaction method.

Accomplishment Description

We have been able to simulate, for the first time, Doppler spectra from nonlinear ocean-surface realizations, as well as scatter from objects near rough surfaces. The simulations have revealed differences between acoustic bottomside cross sections and radar cross sections at the same wavelength. Our simulations have also been used to study the effects of subsurface bubble clouds.

Significance

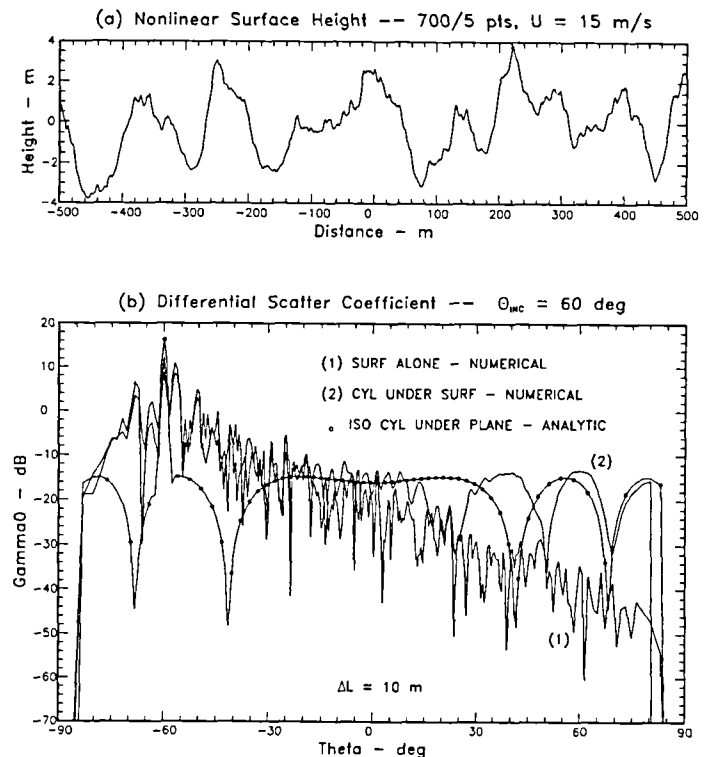
The complexities of nonlinear hydrodynamics, rough-surface scattering, and mutual interaction are sufficiently complex in their own right that few analytic results are available. Combining these effects via numerical simulation is presently the only way to model the full range of observable ocean-surface scatter phenomena.

Future Plans

At the present time, our simulations have been restricted to one-dimensional surface realizations. Although there is much to be learned from such models, we will be implementing some promising schemes that may be fast enough to allow realistic two-dimensional simulations.

Publications

1. Rino, Charles L., and Ngo, Hoc D. "Numerical Simulation of Acoustic Scatter from Subsurface Bubble Clouds." ONT/DARPA Technical Report N00014-87-C-0437, 1989.
2. Rino, Charles L.; Crystal, Thomas L.; Koide, Alan K.; Ngo, Hoc D.; and Guthart, Harold. "Numerical Simulation of Acoustic and EM Scattering from Linear and Nonlinear Ocean Surfaces." Submitted to *Radio Science*, 1990.



Simulation of acoustic bistatic scatter for an isolated nonlinear surface and a cylinder 10 m below the surface reference level. The wavelength is 7.5 m and the source field is incident at 60° from vertical. The surface realization is shown in the top frame. The peaked crests and more rounded troughs are characteristics of ocean surface waves. Note the bistatic enhancements above the level that would be observed for a smooth surface, which is the dotted curve. The mutual interaction method was used to obtain the exact analytic result for an isotropic scatterer above a smooth surface.

Turbulent Reacting Flows

Michael M. Rogers, Principal Investigator
Co-investigators: Robert Moser and Chris Rutland
NASA Ames Research Center

Research Objective

The objective is to generate a direct numerical simulation data base for reacting turbulent flows that will aid in the study of combustion problems by providing an understanding of the effects of turbulence on reaction. Early work will focus on the low-heat-release limit of reaction in incompressible turbulence.

Approach

A direct numerical simulation of the full three-dimensional Navier-Stokes equations is performed with resolution adequate to resolve both the turbulence and the flame zone. Early work is based on reacting passive scalars in an incompressible turbulence, using pseudospectral numerical methods.

Accomplishment Description

Several three-dimensional numerical simulations of temporally evolving, plane mixing layers were generated. The effect of streamwise "rib" vortices on the "rollers" of spanwise vorticity formed by the Kelvin-Helmholtz instability was studied for a variety of low-wave-number initial disturbances. The flows were simulated for up to two pairings of the spanwise rollers with a spanwise domain containing up to four unequal-strength pairs of counter-rotating rib vortices. It was found that despite the lack of any broadband initial disturbances, some of the simulated flows undergo a transition to turbulence. This "mixing transition" is marked by a significant increase in small-scale vortical structures of a highly disorganized nature. The enhanced mixing associated with such a flow results in increased product production, the amount of which is in

agreement with that seen experimentally. The figure shows product concentration at one spanwise location of a mixing layer at completion of the second pairing. The instantaneous flame surface is indicated by the black line. The roughly uniform product concentration in the roller core (figure center) is in line with experimental observations. Simulations required up to 400 Cray Y-MP CPU hours and 70 megawords of memory.

Significance

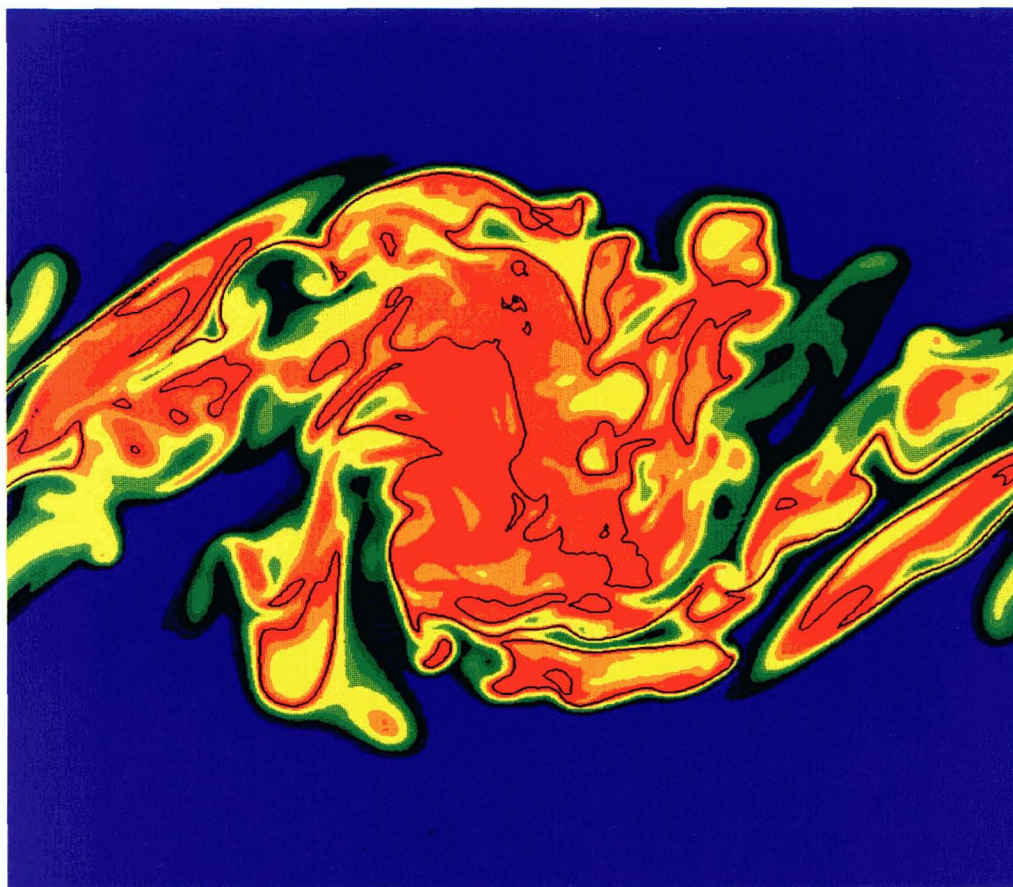
These direct numerical simulations are the first to simulate the transition to turbulence in a plane mixing layer. The mechanisms of this transition are under study and should provide insight into the control of mixing and combustion in free shear flows.

Future Plans

We plan further postprocessing of the numerical data base in an effort to better understand the mechanisms of mixing transition. The effects of a reduced chemical reaction rate will be studied. Long-term goals include simulating reactions with heat release.

Publications

1. Presented at the 1989 Turbulent Shear Flows conference.
2. Presented at the 1990 IUTAM Symposium on Fluid Mechanics of Stirring and Mixing.
3. In preparation for submission to *J. Fluid Mechanics*.



Product concentration at one spanwise location of a mixing layer at completion of the second pairing.

Hydrogen-Air Combustion Simulation in a Pulse Facility

R. Clayton Rogers, Principal Investigator

Co-investigator: Elizabeth H. Weidner

NASA Langley Research Center

Research Objective

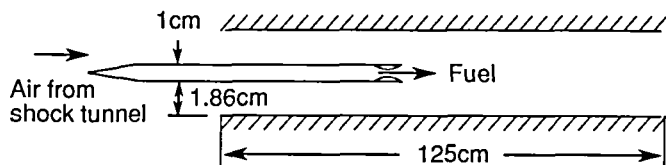
To use computational fluid dynamics (CFD) codes to model the supersonic/hypersonic mixing and combustion of hydrogen in an air flow that is entering a scramjet at conditions that correspond to hypervelocity atmospheric flight.

Approach

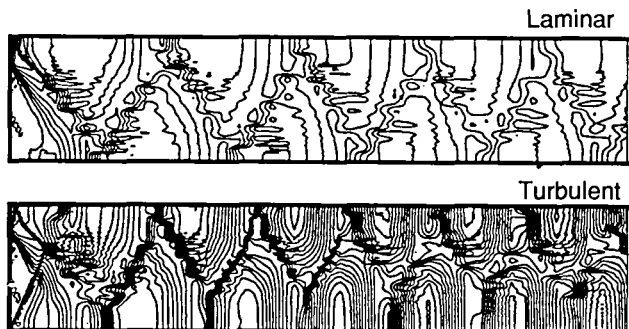
Two- and three-dimensional Navier-Stokes solvers (SPARK) with turbulence, and a full eighteen-reaction, nine-species, H_2 -air, finite-rate combustion model are used to compute the flow through simple combustor configurations that have been tested in the T4 Shock Tunnel at the University of Queensland, Australia. Both full and parabolic versions of the CFD codes are applied, depending on the flow geometry.

Accomplishment Description

The accomplishment during this year's project consisted primarily of calculations made with the parabolized Navier-Stokes (PNS) version of the SPARK code for a combustor configuration, shown schematically in the upper left figure. Some effort was spent learning the PNS code and the techniques for applying it to the quasi-steady flow from a pulse facility. The total duct flow was computed in two parts: from the leading edge of the fuel strut to the strut base; and then patching this flow to begin the solution, with added fuel injection from the strut base, to the duct exit. Results are shown for one test condition with inflow velocity, pressure, and temperature of 3670 m/sec, 21.5 kPa, and 1165 K, respectively. Fuel is injected parallel to the flow at the upper computational



Schematic of experimental combustion duct.



Pressure contours, reacting flow.

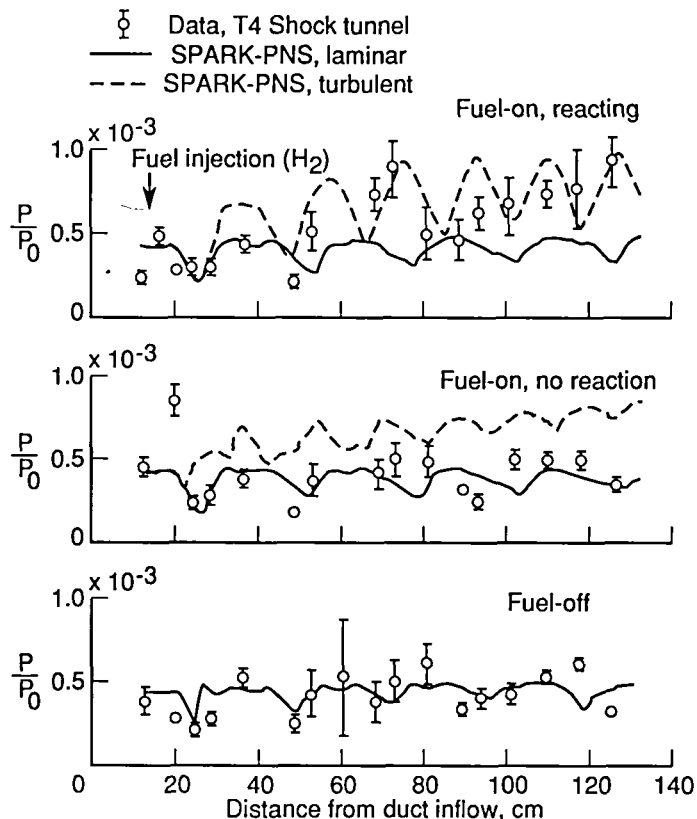
boundary. The lower left figure shows the static pressure field for the reacting flow case for laminar and turbulent flow. The upper computational boundary is the duct centerline from the strut base. The effect of turbulence is to increase the mixing and therefore the combustion, resulting in higher pressures and increased wave angles. This result is also evident in the pressure distributions along the lower wall shown in the figure on the right. The data are from recent tests in the T4 Shock Tunnel at the University of Queensland. In general the agreement is good, considering the uncertainty in some of the data and the turbulence modeling needed for the reacting flow case. These results are preliminary using the PNS code, and the computations are continuing in this year's effort.

Significance

Although preliminary, these results add to our mixing and combustion computational data base for flows at conditions corresponding to hypervelocity flight, and the method provides an alternate means, using the PNS codes, to obtain more rapid calculations.

Future Plans

The use of the SPARK family of codes to analyze and interpret experimental results acquired in pulse facilities at hypervelocity conditions will continue, with the focus on relating the computed results to combustor design parameters of scramjets. These results will be published when a complete set of cases has been computed.



Pressure distributions.

National Aero-Space Plane Inlet Boundary Layer Control, Phase II

William C. Rose, Principal Investigator

NASA Lewis Research Center/Rose Engineering & Research, Inc.

Research Objective

To compute inlet flow fields with novel boundary-layer-removal simulations using the three-dimensional Navier-Stokes equations.

Approach

A modified version of the three-dimensional Kumar internal flow code is run on the Cray-2 with boundary conditions applied to the inlet surface that are representative of those expected to occur from both porous surfaces and portions of surfaces removed.

Accomplishment Description

The code was modified to calculate the flow internal and external to the inlet. A four-block grid using two overlap points at the multiblock interfaces was used to examine the effects of the "correct" interaction between the internal and external flows. Approximately 100 hours of single-processor CPU time was expended, with an average of 16 megawords of run-time

memory. Solutions were obtained for the Mach 5 configuration for conditions representative of those that occur in the NASA Lewis 10- by 10-foot wind tunnel. The figure shows the shaded Mach number contours for half of the inlet on the symmetry plane and four selected crossflow planes. The ramp surface is at the bottom and the cowl is at the top. The multiblock grid is depicted at the last crossflow plane. The side-wall vortical flow spills over the cowl lip as indicated at the third crossflow plane.

Significance

Significant interaction between the captured internal flow and the flow that spills over the side walls and cowl lip (external flow) is expected to occur. The code is now capable of accounting for these interactions.

Future Plans

The multigrid capability will be used to investigate the effect of flow passing through porous and/or cutaway surfaces.



Mach number contours for half of the NASP inlet.

ORIGINAL PAGE
COLOR PHOTOGRAPH

National Aero-Space Plane Inlet Flow Fields

William C. Rose, Principal Investigator

Co-investigator: Edward W. Perkins

NASA Ames Research Center/Rose Engineering & Research, Inc.

Research Objective

To compute the fully three-dimensional flow field within inlets typical of those expected to be used on the National Aero-Space Plane (NASP).

Approach

The existing Kumar three-dimensional internal flow code is run on the Cray-2.

Accomplishment Description

Approximately 300 hours of single-processor CPU time were expended with an average of 16 megawords of run-time memory. Solutions were obtained for a representative Mach 8 inlet model on a grid that was $201 \times 23 \times 61$ for half of the symmetric inlet. The flow conditions were those characteristic of flight with a cool wall. The Mach contours are shown in the figure for the entire flow field and an enlargement. Contours are shown on the symmetry plane (nearest plane) and two off-center planes. Crossflow planes at the cowl lip, near the

throat, and near the exit are shown. Near the center, the flow appears two-dimensional, while near the side wall (far plane) the flow becomes highly three-dimensional. This three-dimensional flow causes significant nonuniformities at the entrance to the combustor.

Significance

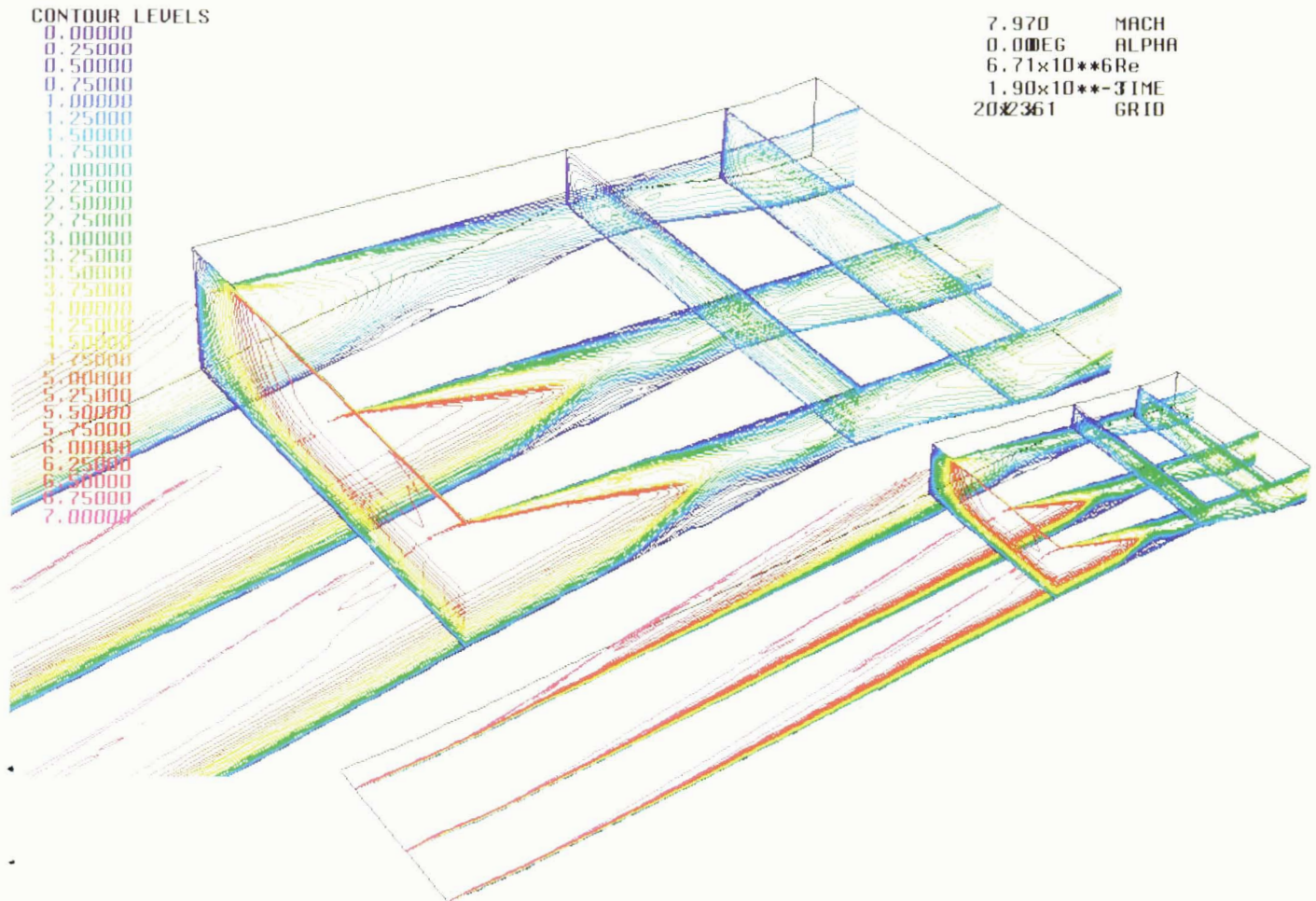
The flow in the two-dimensional models can be computed with a three-dimensional code to quantify the effects imposed on the flow by the addition of the side walls.

Future Plans

Currently this code is being used to solve other three-dimensional inlet and nozzle flow fields expected to be applicable to the NASP vehicle.

Publications

"Comparison of CFD Results and Experimental Data for Scramjet Inlets at $M = 8$ and 18 ." Paper 46, the 7th NASP Symposium, Oct. 1989.



Mach number contours for the NASP inlet; $M = 7.970$, $\alpha = 0.00^\circ$, $\text{Re} = 6.71 \times 10^6$, $\text{time} = 1.90 \times 10^{-3}$, grid size $201 \times 23 \times 61$.

Three-Dimensional Viscous Flow in High-Speed Inlets

William C. Rose, Principal Investigator

Co-investigator: Edward W. Perkins

NASA Ames Research Center/Rose Engineering & Research, Inc.

Research Objective

To compute inlet flow fields with three-dimensional Navier-Stokes equations in order to investigate fully three-dimensional flows and shock-wave effects on inlet distortion.

Approach

The existing three-dimensional Kumar code is run on the Cray-2.

Accomplishment Description

Approximately 50 hours of single-processor CPU time were expended, with an average of 16 megawords of run-time memory. Solutions were obtained on a $101 \times 63 \times 41$ grid for a three-dimensional side-wall compression inlet that had a contraction causing swept shock waves to exist within the inlet in the presence of a thick, oncoming boundary layer. The results

indicate a distorted outflow condition with large variations in Mach number, as shown in the attached figure.

Significance

With the large run-time memory available, solutions to internal flow problems representative of generic high-speed inlets can be obtained with sufficient resolution to demonstrate the potential for distorted flow fields exiting the inlet.

Future Plans

The code is being used to solve other fully three-dimensional inlet flow fields with swept shock-wave/boundary-layer interactions applicable to high-Mach number flows.

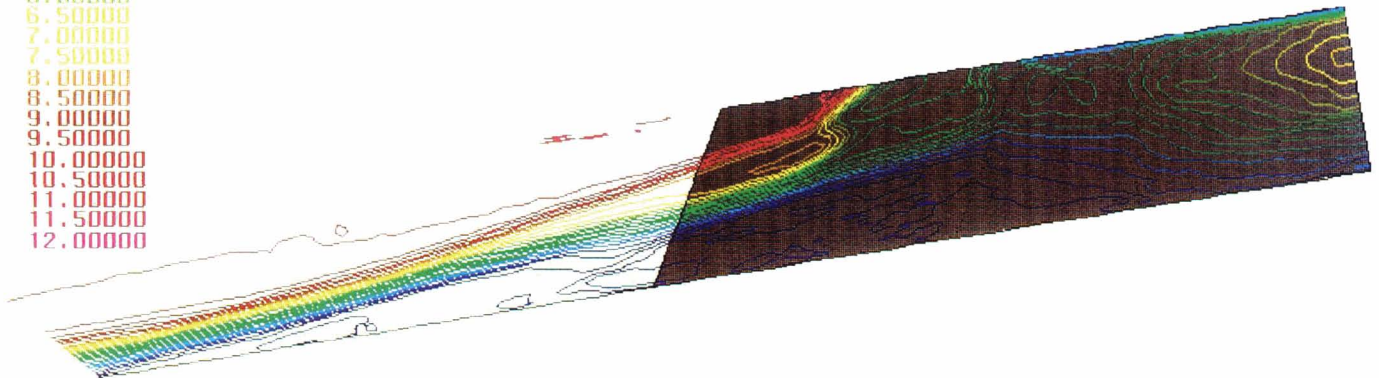
Publications

"Analysis of the Flow in Sidewall Compression Inlets at Hypersonic Mach Numbers." NASP Contractor Report 1064, Oct. 1989.

CONTOUR LEVELS

0.00000
0.50000
1.00000
1.50000
2.00000
2.50000
3.00000
3.50000
4.00000
4.50000
5.00000
5.50000
6.00000
6.50000
7.00000
7.50000
8.00000
8.50000
9.00000
9.50000
10.00000
10.50000
11.00000
11.50000
12.00000

17.700 MACH
0.000 DEG ALPHA
 $0.35 \times 10^{**8}$ Re
 $1.34 \times 10^{**-3}$ TIME
101x63x41 GRID



Mach number contours for an SWC inlet on a 4° wedge; $M = 17.700$, $\alpha = 0.00^\circ$, $Re = 0.35 \times 10^8$, $time = 1.34 \times 10^{-3}$, grid size $101 \times 63 \times 41$.

Advanced Computational Materials

M. B. Salamon, Principal Investigator

Co-investigators: R. Averback, Y.-C. Chang, J. Kogut, R. Martin, and P. Wolynes

University of Illinois, Urbana-Champaign

Research Objective

The program seeks new algorithmic approaches to problems in materials research. New methods for quantum problems and new approaches to statistical mechanics problems are the main focuses of the NAS work. This effort is sponsored by the Materials Laboratory Program of the National Science Foundation.

Approach

Monte Carlo methods, a combination of local-density and molecular dynamics techniques, and the embedded atom method are among the approaches used.

Accomplishment Description

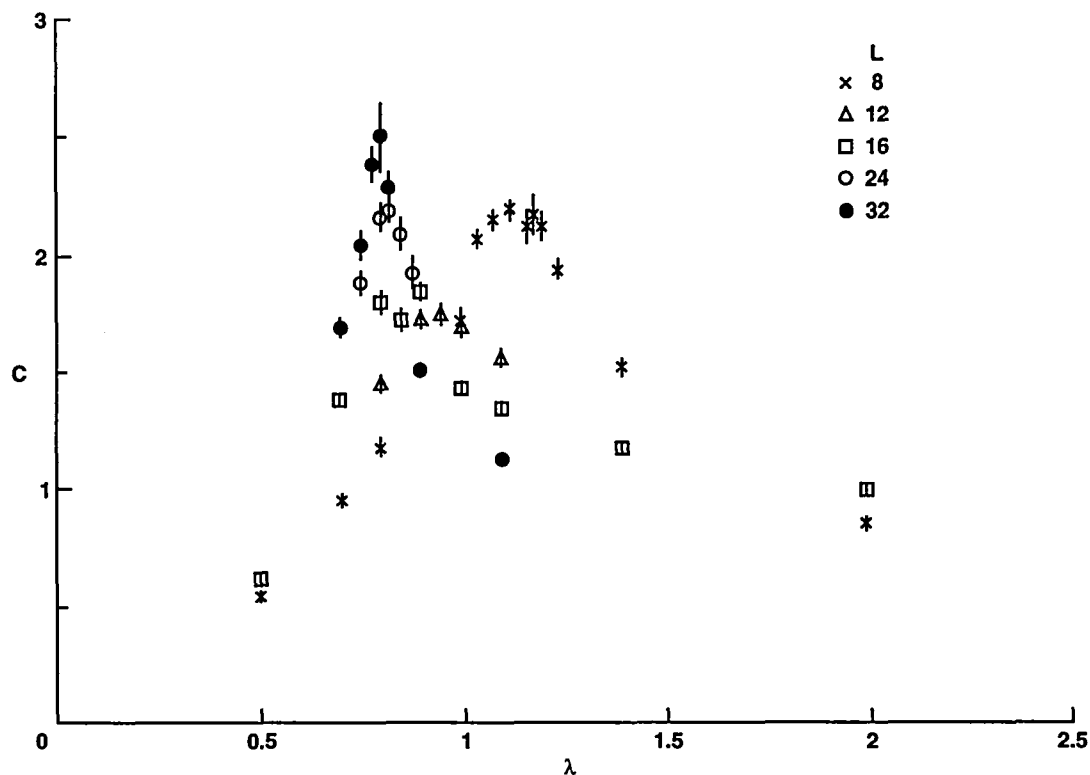
Martin's group developed new local-density methods that, combined with molecular dynamics, are providing a description of liquid carbon at high temperatures and pressures. Renken, with Kogut, applied scaling laws to the specific heat, which has a peak as a function of surface curvature, to confirm that the crumpling transition is of second order with a correlation-length exponent of 0.78(2). The band structure of semiconductor wires was studied by Chang's group. This study has just begun at Ames. Averback simulated the collision of 13- and 92-atom clusters with crystal surfaces. He found that when the cluster velocity exceeds the sound velocity of the crystal, the collisions approximate individual atom collisions, while at lower velocities macroscopic descriptions are adequate.

Future Plans

The researchers on the project plan to extend the development work to actual production runs, but find the turn-around time excessive. Renken will try to compute the central charge of his two-dimensional critical theory. Averback will study clusters with larger energies to investigate "thermal spike" effects. Chang's preliminary work on quantum wires and multilayers will continue.

Publications

1. Averback, R. S. "Ion Beam Mixing at Metal Surfaces." Invited paper, presented at the American Physical Society Spring Meeting, 1990.
2. Hsieh, H., and Averback, R. S. "Molecular Dynamics Investigation of Cluster Beam Deposition." To be submitted to *Phys. Rev. B*.
3. Averback, R. S.; Hsieh, H.; Diaz de la Rubia, T.; and Benedek, R. "Interactions of Energetic Particles and Clusters with Solids." Invited paper, presented at the 7th International Conference on Ion Beam Modifications of Materials, Knoxville, TN, Oct. 1990.
4. Renken, Ray L., and Kogut, John B. "Scaling Behavior at the Crumpling Transition."
5. Renken, Ray L., and Kogut, John B. "The Central Charge of the Crumpling Transition."



The specific heat as a function of the extrinsic curvature coupling for $L = 8$ (x), $L = 12$ (triangles), $L = 16$ (squares), $L = 24$ (empty circles), $L = 32$ (filled circles).

Computational Methods for Rotor-Blade Drag Prediction

Matthew T. Scott, Principal Investigator
Bell Helicopter Textron, Inc.

Research Objective

The objective of this project is to examine the predictive capabilities of a hierarchical assortment of numerical methods for the computation of airflows about helicopter rotor blades. In particular, measurement of the various components of drag on rotor blades is being emphasized.

Approach

Three-dimensional Navier-Stokes codes utilizing both rotating and nonrotating coordinate frames attached to the rotor blade are used to predict drag levels along rotor blades. An unsteady three-dimensional full-potential code in rotating coordinates is used to provide inviscid drag levels. A comparison of results from the codes allows an assessment of the relative magnitudes of the drag components that are due to viscosity, wake induction, and algorithm.

Accomplishment Description

Existing three-dimensional viscous codes were modified to facilitate computations of rotor blades in nonrotating coordinate systems as well as rotating ones. Boundary conditions for one viscous solver were rewritten so the results could be compared with existing wind tunnel data. Another viscous code was modified to disable rotational motion of the coordinates and thus to provide algorithmic correlation with the first solver. The inviscid rotor code was used to provide baseline spanwise drag levels on both proprietary and general-use rotor planforms and also to provide insight into unsteady shock collapse on the advancing part of the disk. The accompanying figure shows some correlations made on a solution

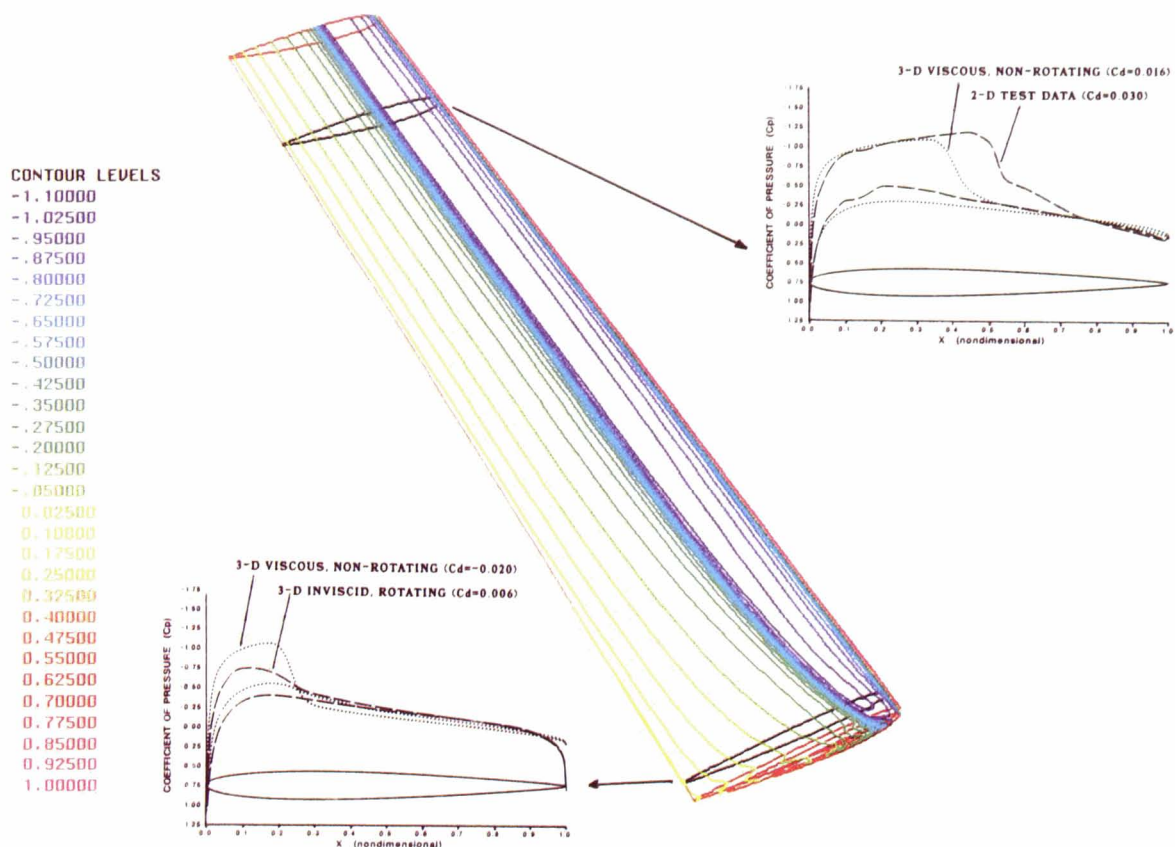
for a NACA 0012 rotor at transonic conditions and yawed relative to the free stream. Away from the tip region, this code shows a shock location approximately 12% forward of that shown by two-dimensional test data. Other test data (Harris, NASA TM-81927, 1981) show the shock location to be between these two values, at 46% chord. Drag values differ by 14 counts; this is mostly attributable to the difference in shock location. At the tip, the shock has moved forward on the blade. In nonrotating coordinates, this movement results in a net thrust on that section. However, inviscid results in rotating coordinates show that the shock is almost nonexistent and that the section still produces drag. Further research tested the effect of yaw angle on shock location and wake model on required rotor power.

Significance

Requirements for modern high-speed helicopters include fast rates of climb, improved performance, and high degrees of maneuverability. By reducing the levels of certain components of drag on the blade, more power is made available to the rotor for these tasks. Accurate assessment of blade drag in maneuvers, transonic flight, or at high α is crucial to rotorcraft predictive and analytic methodology.

Future Plans

Analyses of the drag components that are due to azimuthal blade position, wake induction, and three-dimensional effects will be continued. Viscous solutions will be provided to correlate with wind tunnel tests of high-speed tip planforms.



Pressure contours on a NACA 0012 rotor blade; $M = 0.80$, $\alpha = 2.40^\circ$, yaw angle = 20° forward, $Re = 3.7 \times 10^6$.

Direct Simulation of High-Speed Mixing Layers With and Without Chemical Heat Release

Balu Sekar, Principal Investigator

Co-investigators: H. S. Mukunda and Mark H. Carpenter

NASA Langley Research Center

Research Objective

To examine the effects of convective Mach number, disturbance levels, initial profiles, Reynolds number, and chemical heat release on the structure of a high-speed mixing layer.

Approach

The SPARK2D code is used to perform a direct numerical simulation of the high-speed two-dimensional mixing layer. The numerical algorithm used is the dissipative compact parameter scheme, which has a formal accuracy of $O(\Delta t^2, \Delta x^4)$.

Accomplishment Description

High-speed mixing-layer calculations were made at convective Mach numbers of 0.38 and 0.76 for a number of initial disturbance levels, for two initial profiles, and for a range of Reynolds numbers, using the SPARK2D code. Fine grids were used to ensure that all the relevant length scales were resolved. From the calculations, the following conclusions were drawn: (1) Assumed hyperbolic tangent profiles need large disturbance levels to make the flow transitional. (2) The concept of a convective Mach number needs review at high convective Mach numbers because the structures dilate significantly. (3) The growth rates obtained from the time-averaged data compare reasonably well with existing experimental data, illustrating the compressibility effects. (4) The significant role of heat release is to reduce the growth rate of the mixing layer by about 5 to 7% and to reduce the convective speed of the structures by about 10% at $M_c = 0.38$. Preliminary results indicate that increasing the convective Mach number will reduce the effect of heat release.

(5) Changes in density in the flow result from temperature, pressure, and composition variations in the compressible case, unlike those in incompressible flows, in which temperature and composition are the dominating factors. (6) The reduction in Reynolds stresses and kinetic energy of fluctuations, as well as the reduction of entrainment with heat release, are akin to those in incompressible flows, but to a much lesser extent. In view of the weaker role of heat release, it is reasonable to conclude that it is useful to concentrate on nonreacting flows for the mixing-related issues. Heat release is likely to provide only a small perturbation to many nonreacting flows.

Significance

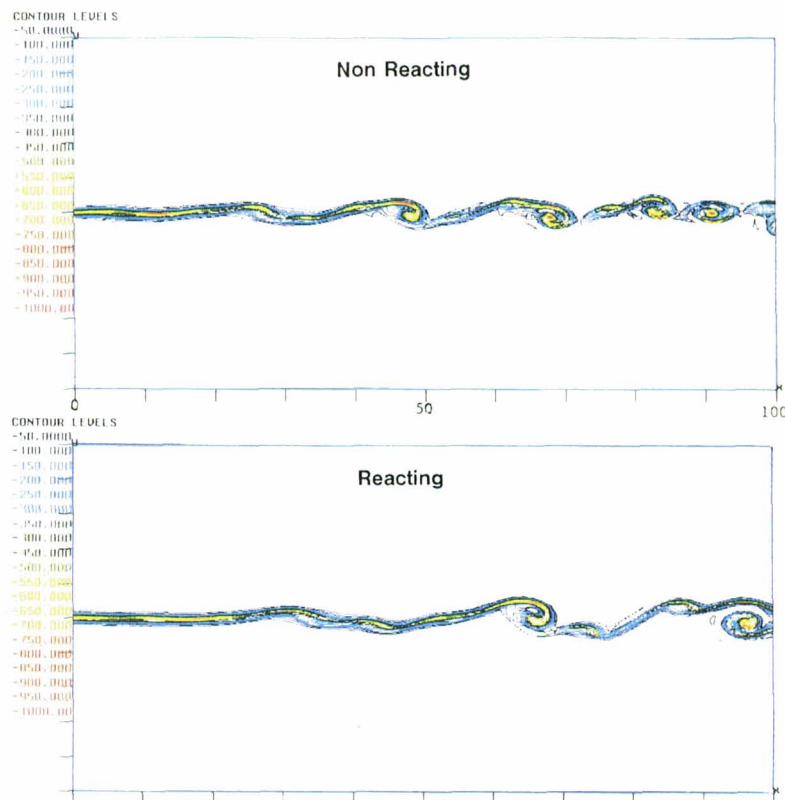
Direct numerical simulation of high-speed mixing layers has improved the understanding of fuel-air mixing in scramjet combustors. Current studies have facilitated the determination of several important techniques for producing high levels of mixing and combustion efficiency in a scramjet combustor.

Future Plans

Flow conditions exist for which the above conclusions may be only partially valid. Further study of these issues as well as a complete study of three-dimensional nonreacting and chemically reacting mixing layers is necessary.

Publications

Sekar, Balu; Mukunda, H. S.; and Carpenter, M. H. "The Direct Simulation of High Speed Mixing Layer Without and With Chemical Heat Release." Presented at the Computational Fluid Dynamics Symposium on Aeropropulsion, NASA Lewis Research Center, Cleveland, OH, Apr. 1990.



Vorticity contours for a nonreacting and a reacting H_2 -air mixing layer; $M_c = 0.38$, domain size 100×50 mm, grid size 201×125 .

Ionized Flow Around a Reentry Vehicle

John V. Shebalin, Principal Investigator
NASA Langley Research Center

Research Objective

The objective of this work is to numerically simulate plasma electrodynamic processes in the ionized flow that occurs around an aerospace vehicle during high-Mach number reentry. This will provide the ability to study the effects of electromagnetic (EM) stresses on spacecraft design and performance, on flow-field modeling, and on EM wave propagation through the plasma surrounding a hypervelocity reentry vehicle.

Approach

A two-dimensional, compressible magnetohydrodynamic (MHD) spectral code, with a polytropic isentropic equation of state, was used to study the growth and homogeneous dynamics of the self-magnetic field. A Fourier collocation time domain (FCTD) method was created to study EM propagation in a plasma medium.

Accomplishment Description

(1) A Fourier spectral transform method code was used to study magnetic field amplification in forced MHD turbulence. Initial conditions consisted of an external mean magnetic field and a negligible turbulent magnetic field; it was found that although the final energy in the turbulent magnetic field depended directly on the initial external mean field strength, the actual amplification of the turbulent magnetic field varied inversely with the mean field strength. Also, compressibility appeared to have only a marginal effect in mediating the transfer of kinetic into magnetic energy. (2) A new, two-dimensional, EM propagation code using an FCTD method was created. The advantage of this method over the popular finite-difference time domain (FDTD) method lies in an accuracy that is greatly increased for a relatively small increase in computation time, while the ease of use of FDTD is maintained. In these initial investigations, both of these efforts traded relatively small grid sizes (64×64 and 128×128) for several dozen separate simulations, each separate simulation lasting 3 to 4 CPU hours and using 1 to 2 megawords of memory.

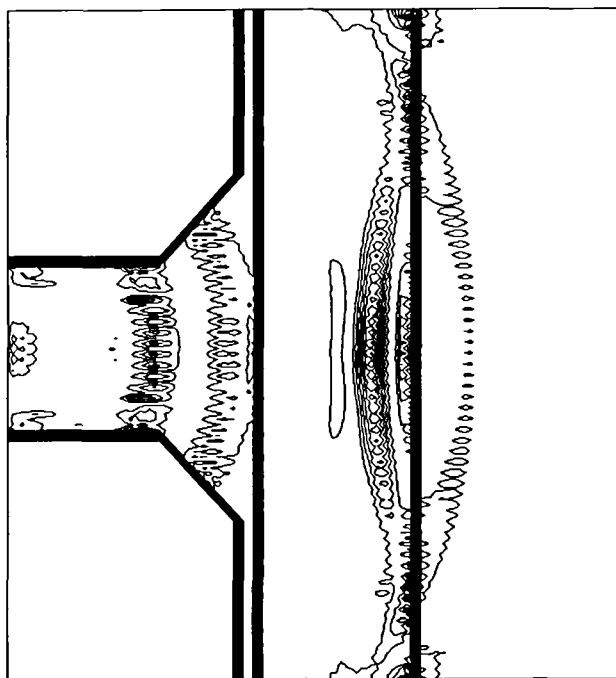
Significance

As NASA moves further into the realm of hypersonic flight, the need to adequately understand ionized aerodynamic flow (i.e., plasma electrodynamics) in addition to aerothermodynamics becomes more critical. Such flow, and the novel features it may contain, is associated with atmospheric reentry, flight, and aeromaneuvering. Aerospace systems that generate ionized flows include the planned Aeroassist Flight Experiment vehicle, future aeroassisted orbital transfer vehicles, the

National Aero-Space Plane, and tethered satellites. Furthermore, an understanding of ionized flow fields may shed some light on those unexpected phenomena encountered during shuttle reentry: hypersonic pitching moment and jet-aileron interactions. Plasma processes are also expected to be important in hypervelocity shocks.

Future Plans

In the coming year (NAS 1990-91), the primary planned objectives are an investigation into magnetic field and electric current generation occurring in plasma shock fronts, and further investigations into EM wave propagation in inhomogeneous plasmas. Three-dimensional simulations are planned.



Regional energy = 0.082 = 0.171 = 0.053
Total energy = 0.306 at 0.32 nsec

A 12.5-GHz single-wavelength sinusoidal pulse propagating out of a microwave horn, through a thermal tile, and into an over-dense (reentry) plasma; total energy = 0.306 at 0.32 nsec.

Afterbody Aerothermodynamic Studies for the Hypersonic Glide Vehicle

Peter K. Shih, Principal Investigator
Co-investigator: Che-Shing Kang
General Dynamics, Convair Division

Research Objective

To develop a computational fluid dynamics (CFD)-based analytical method for the calculation of the aerothermodynamic environments of the Hypersonic Glide Vehicle (HGV), with the emphasis on the aftbody flow field.

Approach

The technical approach to this study is to validate existing CFD techniques against the HGV test data. In 1986, heat transfer and pressure data were obtained from the Calspan shock tunnel for a 0.3-scale model of the HGV. These data were compared with the PARC3D results. The 0.3-scale model was heavily instrumented in the aftbody region, which consisted of the fuselage, wing, tip fin and elevon surface. A good test case was thus provided for the PARC3D code.

Accomplishment Description

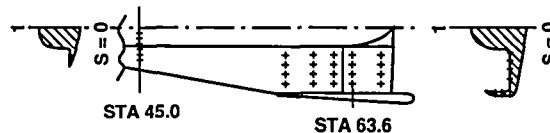
The full Navier-Stokes (FNS) solutions for the 0.3-scale HGV model were obtained using the PARC3D code under various Calspan test conditions, including both laminar and turbulent flows at zero or moderate angles of attack. In general, both laminar and turbulent FNS results agreed well with Calspan test data. To demonstrate the results, the turbulent FNS solutions, using the Baldwin-Lomax turbulence model, are presented in the accompanying figure. A heat flux comparison along the periphery of two axial locations of the HGV model is shown for 0° and 8.0° angle of attack. At Station 45, both laminar and turbulent results are plotted since the test data show the characteristic of boundary layer transition, whereas at Station 63.6, the fully turbulent solutions seem to fit the test data satisfactorily. A typical PARC3D computation for the HGV whole-body takes 20 Cray-2 hours and 10 megawords of memory.

Significance

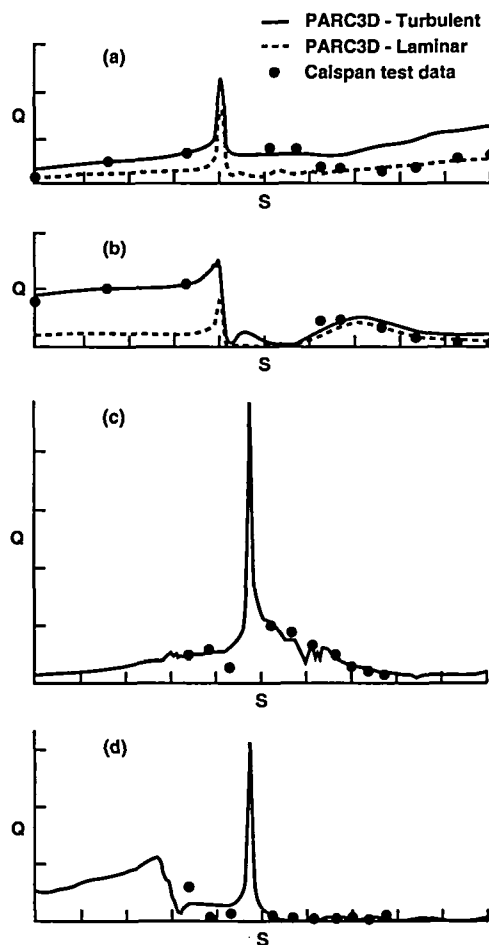
The design of future hypersonic vehicles requires a complete understanding of the aerothermodynamic environment, which is especially severe because of the long endoatmospheric mission durations desired and the high lift-to-drag ratio necessary for maneuvers. Because of the lack of applicable test facilities, CFD solutions are particularly required in designing these vehicles.

Future Plans

The 1990 objectives are to (1) develop dynamic gridding techniques for PARC3D applications, (2) compare turbulent models to determine the effects on heating data prediction, and (3) model the flow field near a deflected elevon in



Placement of the instrumentation on the 0.3-scale HGV model.



Comparison of heat transfer data from the PARC3D code and the Calspan wind tunnel tests; $M = 11.5$, $Re = 10.0 \times 10^6/\text{ft}$.
(a) Station 45.0, $\alpha = 0.0^\circ$; (b) Station 45.0, $\alpha = 8.0^\circ$;
(c) Station 63.6, $\alpha = 0.0^\circ$; (d) Station 63.6, $\alpha = 8.0^\circ$.

Advanced Numerical Algorithms for Chemically Reacting Flows

Jian-Shun Shuen, Principal Investigator
Sverdrup Technology, Inc./NASA Lewis Research Center

Research Objective

The objective of this work is to develop a new procedure that combines the accuracy of modern upwind schemes and the efficiency of the lower-upper (LU) factorization scheme for high-speed reacting flows. The resulting code has to be efficient and robust for flows involving high Mach numbers, strong shocks, and large zones of flow separation.

Approach

An efficient upwind method for solving the Navier-Stokes equations with nonequilibrium, frozen, and perfect-gas chemistry models is developed to calculate high-Mach number, high-enthalpy flows. The method uses either Roe or van Leer flux splitting for inviscid terms and central differencing for viscous terms in the explicit operator, and Steger-Warming splitting and LU factorization for the implicit operator.

Accomplishment Description

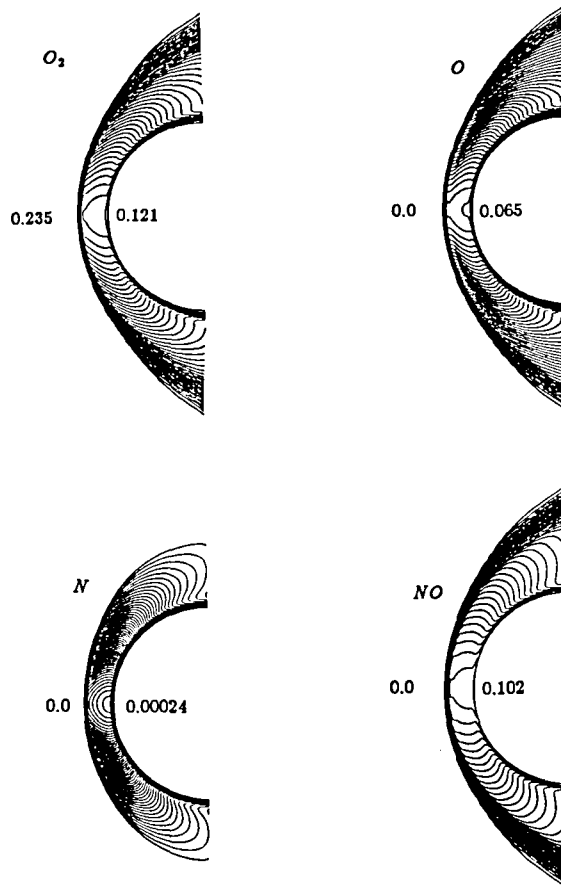
An implicit, upwind, two-dimensional Navier-Stokes code was developed for high-speed reacting flow calculations. The code uses upwind, flux-splitting total-variation-diminishing schemes and LU factorization. The flow and chemistry are fully implicit and fully coupled to achieve efficiency and robustness. Sophisticated thermodynamic and transport property models are used for realistic evaluation of thermo-physical properties at high temperatures. A series of numerical tests for a variety of flow configurations at varying Mach numbers and total temperatures was conducted to study the effects of real-gas chemistry on the structure of these flow fields. The test cases demonstrate the good shock-capturing capability as well as the efficiency and robustness of the code. The accompanying figure shows the species mass fraction contours for flows over a half-cylinder at free-stream $T = 600$ K and $M = 7.0$. A typical chemical nonequilibrium calculation with a 100×100 grid needs about 1.5 Cray Y-MP hours and 3 megawords of memory.

Significance

The numerical method and computer code developed in this work can be used to simulate both external and internal flows for the proposed aerospace vehicles in actual flight conditions.

Future Plans

The code currently uses an air dissociation model for the nonequilibrium chemistry. It is planned to extend the chemical kinetics to include both the air disassociation/ionization and hydrogen-air reactors.



Species mass fraction contours for flow over a half-cylinder;
 $M_\infty = 7.0$.

Ignition and Flame Spread above Liquid Fuel Pools: Gravity Effects

William A. Sirignano, Principal Investigator

Co-investigators: Fanghei Tsau and David N. Schiller

University of California, Irvine

Research Objective

The objective of this research is to understand the basic mechanisms of buoyancy and/or surface-tension-driven flows and thereby provide adequate engineering scaling laws to guide experimental designs for reduced-gravity environments. Both two- and three-dimensional computational capabilities are required to simulate numerically an unevenly heated two-phase flow inside a cylindrical enclosure. The three-dimensional study is needed when a variable acceleration field is present.

Approach

Two- and three-dimensional Navier-Stokes computer codes with variable fluid properties are used to predict the temperature and flow fields inside the cylindrical enclosure. The numerical codes use a hybrid differencing scheme and a semi-implicit solution procedure.

Accomplishment Description

The Cray hours allocated were used for three-dimensional studies, so the current discussion focuses on the results gained from them. Numerical simulations were conducted for a single-phase flow with a prescribed periodic acceleration field. The conventional Boussinesq approximation for changes of fluid properties was found to overpredict the heat transfer rate and to have a stronger tendency to respond to external agitations. The variable acceleration field (the so-called g-jitter) alters the heat transfer and flow characteristics significantly. For example, the heat transfer rate becomes periodic and the interaction between the thermal field and the flow field also becomes stronger. Strong three-dimensional effects result in complicated cellular flow structure and larger advection speed. The study also finds that reducing the total gravity level is better than just terminating the external g-jittering to restore

a perturbed system. Different fluids have different post-jitter responses closely related to the viscosities of the working fluids. The external acceleration components that are perpendicular to the imposed thermal gradient have positive effects in increasing the heat transfer, but the role of the component parallel to the imposed thermal gradient is somewhat ambiguous. A typical variable-property run with simulation of up to two cycles of gravity modulation requires 10 Cray-2 hours and 5 megawords of memory.

Significance

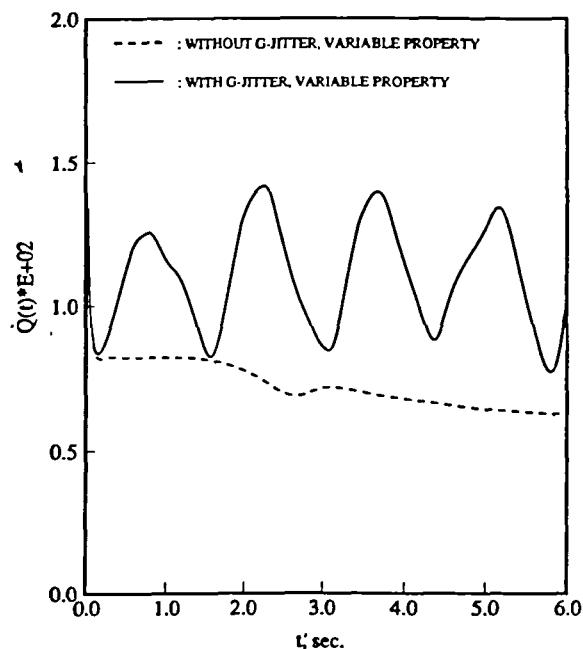
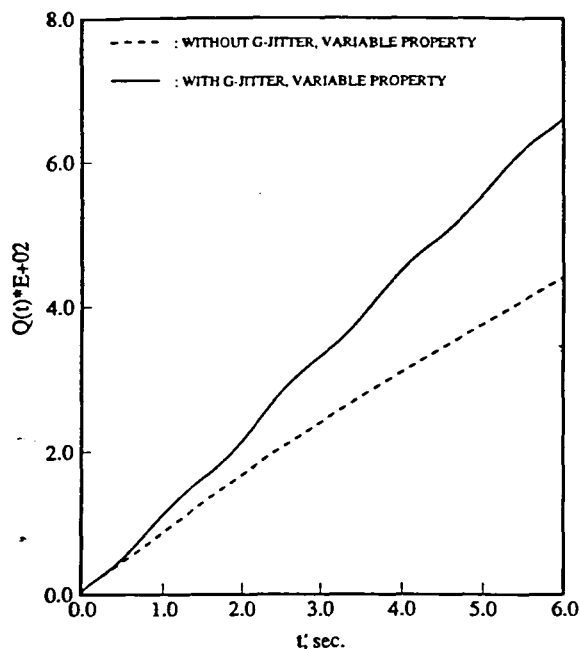
This research relates to hazardous fire at Earth-gravity conditions and aboard space vehicles at reduced-gravity conditions where a pool of flammable fuel is in the vicinity of an initial fire. Understanding the interaction between the buoyancy and the surface tension (along the surface of the fuel pool as a result of nonuniform heating) in various gravitational conditions can lead to the understanding of mechanisms for ignition and flame spread. Preventative measures can then be set up to avoid irreversible damages. Knowledge of the g-jitter effects can also be beneficial to material processes and can increase the success rate of space experiments.

Future Plans

The three-dimensional code will be further implemented to conduct two-phase calculations. Thermal-capillary motions will be studied first under different gravity levels with various fluids. The knowledge and capabilities accumulated will then be used to investigate the two-phase flow with g-jitter effects.

Publications

Tsau, F.; Elghobashi, S.; and Sirignano, W. A. "Effects of G-Jitter on a Thermal, Buoyant Flow." Presented in the 28th Aerospace Sciences Meeting, Reno, NV, Jan. 1990.



Thermal energy and its rate histories for the variable-property cases with and without g-jitter.

Transition and Turbulence in Three-Dimensional Flows

Philippe R. Spalart, Principal Investigator
NASA Ames Research Center

Research Objective

The objective is to use direct numerical simulations of selected transitional and turbulent boundary layers to contribute to the understanding and prediction of aerodynamic flows. Recent efforts cover a range of more complex flows—for instance, spatially developing, three-dimensional, and separating flows.

Approach

We use a well-established spectral solver of the incompressible, three-dimensional, time-dependent, unaveraged Navier-Stokes equations over a flat plate. It can now treat spatially developing flows thanks to the “fringe” method. Each study involves choices of boundary conditions (pressure gradient), initial disturbances, and analysis procedures.

Accomplishment Description

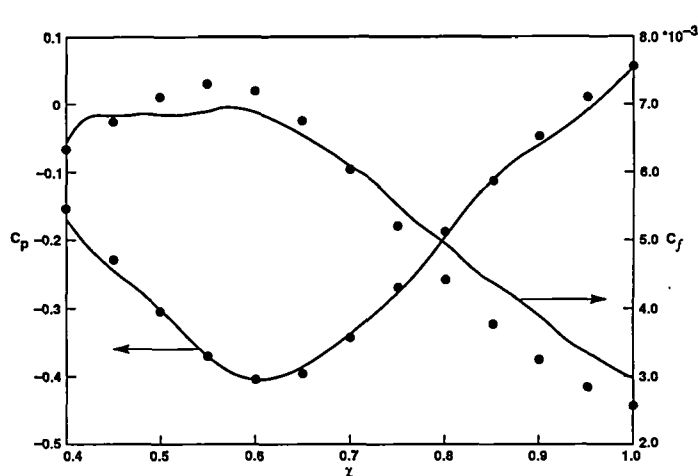
(1) Direct simulations of crossflow vortices in a truly spatially developing, laminar boundary layer were conducted for the first time. (2) Direct simulations of a truly spatially developing, turbulent boundary layer were conducted for the first time and compared with a companion experiment conducted at Ames by Watmuff.

Significance

Item (1) accurately provides the growth rates of vortices, which often cause transition on swept wings, without the controversial parallel-flow approximation. Item (2) includes all effects of the pressure gradient on the turbulence; experiment and simulation are closely matched, including the pressure distribution and the Reynolds number, and the results compared in depth, resulting in a high level of confidence (see figure).

Future Plans

For the Watmuff flow we will obtain statistics, analyze the results from a theoretical and modeling point of view, and publish it. We are also simulating a three-dimensional boundary layer with pressure gradient, and a separation bubble.



Pressure and skin friction for a spatially developing, turbulent boundary layer; • experiment, —simulation.

Magma Chamber Convection

Frank J. Spera, Principal Investigator
University of California, Santa Barbara

Research Objective

The goal of this work is to understand the dynamics of viscous flows in which two sources of buoyancy (chemical and thermal) are present. Nonlinearities inherent in the convection-diffusion transport equations as well as in the rheological constitutive law give rise to complex flows in time and space. We wish to understand the dynamical basis of these complexities.

Approach

Two codes, a finite-element one and a primitive variable one valid for both two- and three-dimensional situations, were used to study single-phase and multiple-phase (e.g., solidification plus convection) mass and energy transport in doubly diffusive systems.

Accomplishment Description

A parametric study was used to determine the conditions in parameter space (Ra , Rs , Le , Pr) with and without crystallization, such that the computational domain becomes chemically heterogeneous starting from a constant composition field.

Significance

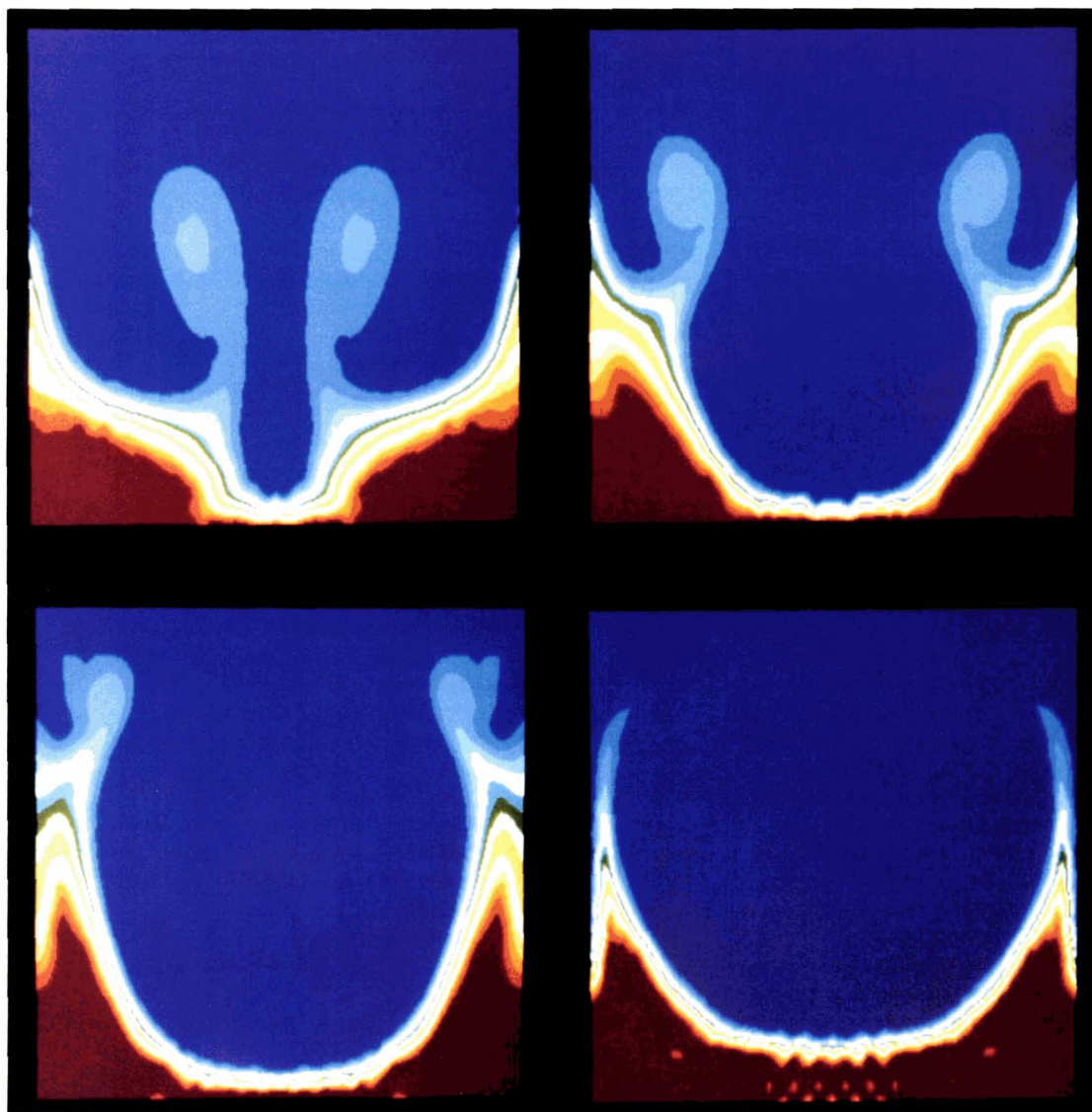
Magma chambers within the Earth's crust commonly show evidence of in situ compositional zonations. These simulations show in detail how chemical heterogeneity can develop from magma bodies that initially are unzoned but that mix actively by buoyancy-driven convection.

Future Plans

The codes will be further altered to incorporate more realistically the physics of convection with phase change. Specifically, more attention will be focused on the two-phase region (the mush).

Publications

1. Spera, Frank J.; Oldenburg, Curtis M.; and Yuen, David A. "Magma Zonation: Effects of Chemical Buoyancy and Diffusion." *Geophysical Research Letters* 16, no. 12 (1989): 1387-1390.
2. Trial, Alain F., and Spera, Frank J. "Mechanisms for the Generation of Compositional Heterogeneities in Magma Chambers." *Geological Society of America Bulletin* 102 (1990): 353-367.



Magma chamber convection.

ORIGINAL PAGE
COLOR PHOTOGRAPH

National Aero-Space Plane Engine Corner-Flow

B. M. Steinetz, Principal Investigator

Co-investigators: W. J. Coirier and Mike Tong

NASA Lewis Research Center/Sverdrup Technology Inc.

Research Objective

The objective of this work is to determine the flow field and heat transfer in a panel-edge seal system of advanced hypersonic engines such as the National Aero-Space Plane (NASP). These seals must prevent the extremely hot, pressurized, engine flow-path gases from escaping through the interfaces between the articulating engine panels and the stationary splitter walls. Accurately estimating the seal heat fluxes and the ambient seal channel temperatures is a key to determining the seal-coolant resources that are required to prevent seal over-temperatures. Three-dimensional flow analyses are required in this study to assess the effects of the engine corner and the seal recess depth on the actual heat flux into the seal.

Approach

A three-dimensional Navier-Stokes code, PARC-3D, was used to investigate the flow field and heat transfer in a panel-edge seal configuration. To improve the convergence rate for these studies, a hybrid lower-upper symmetric Gauss-Seidel (LU-SGS) algorithm was implemented.

Accomplishment Description

An example of the type of simulations possible with this advanced capability is shown in the accompanying figures. In this study, Mach 10 air is flowing in the engine inlet, which is modeled by horizontal and vertical engine panels. The panel-edge seal is slightly recessed below the horizontal panel. The figure on the left shows the flow velocities at four engine-inlet positions; pink is the high-speed flow and blue is the low-speed flow. Note that the seal-channel flow quickly deceler-

ates as the boundary layer grows, limiting the heat flux into the seal. The three-dimensional character of the flow is demonstrated by the particle traces shown in the right-hand figure. Based on the flow calculations made, heat fluxes to the seal were determined and subsequently used for thermal stress analyses of the panel-edge seal system. To improve overall efficiency of the code, an LU-SGS algorithm was implemented for the approximate Newton iterations; this also required less user interaction to achieve acceptable convergence rates. An efficient method to account for limited real-gas effects was also made available to PARC-3D. A typical computer run required 8 megawords of memory and 2 CPU hours on the Cray-2.

Significance

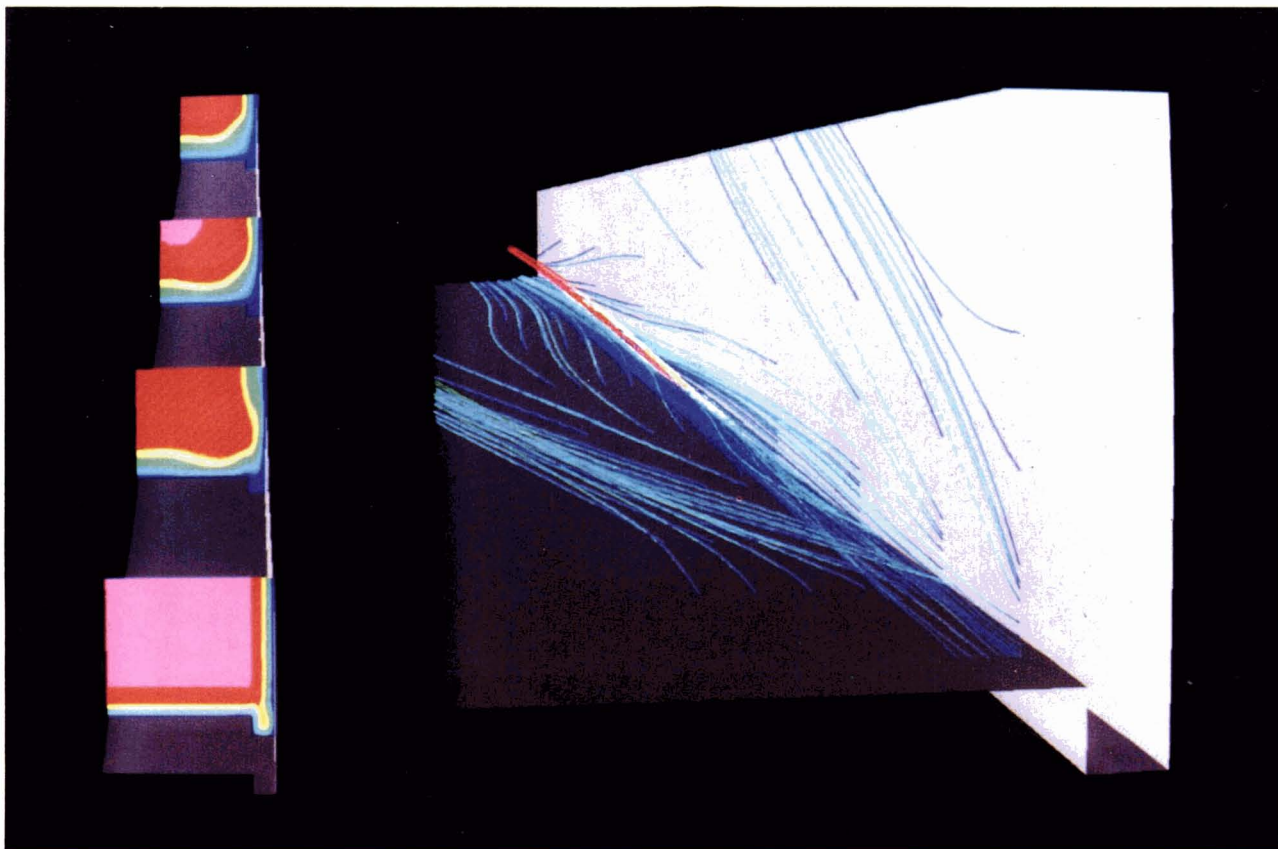
The panel-edge seal system is a critical mechanical system in an advanced hypersonic engine. These seals must prevent the extremely hot, pressurized, engine flow-path gases from escaping past the movable panels. Successful simulation of the fluid dynamic and heat transfer phenomena in and around the seal system is crucial to the design of the hypersonic engine.

Future Plans

Future work will investigate the fluid dynamic and heat transfer phenomena of the seal system with various gap aspect ratios (e.g., exposed seal-width/recess-depth).

Publications

Coirier, W. J. "High Speed Corner and Gap-Seal Computations Using an LU-SGS Scheme." AIAA Paper 89-2669, July 1989.



Hypersonic gap seal simulation. (Left) Flow velocities at four engine-inlet positions. (Right) Particle flows.

Computational Fluid Dynamics Prediction of Advanced Rotor Performance

Roger C. Strawn, Principal Investigator

Co-investigators: John Bridgeman, K. Ramachandran, and Frank Caradonna

U.S. Army Aeroflightdynamics Directorate—AVSCOM/NASA Ames Research Center

Research Objective

This project is aimed at developing methods for predicting the aerodynamic loads on helicopter rotor blades. Primary areas of investigation are unsteady transonic flow, viscous contributions to drag and pitching moment, and the influence of the rotor wake. The development of computational fluid dynamics codes for immediate use as rotor design tools is emphasized.

Approach

Three-dimensional unsteady potential codes are used to compute the inviscid flow. The full-potential code is modified with an entropy correction term to give Euler shock-jump behavior. A three-dimensional, unsteady boundary layer code is coupled to the inviscid solution to compute viscous drag. The rotor wake is modeled with a combined Eulerian-Lagrangian scheme to compute the rotor and free-wake solutions simultaneously.

Accomplishment Description

The Helix-1 hover performance prediction code was extended to obtain the compressible free-wake flow about a lifting rotor-body. The method is an extension of a unique vorticity embedded full-potential method. It was used to predict the performance of a full-scale rotor in partial ground effect with the rotor mounted on a large pylon. In a separate investigation, a three-dimensional, finite-difference boundary layer scheme was coupled to the full-potential rotor code (FPR) to predict drag and torque for rectangular and swept-tip rotor blades. Also, the FPR code was coupled to the CAMRAD/JA rotor performance code to model a swept-tip Puma helicopter rotor in high-speed forward flight. Results compared well with extensive flight-test measurements. Typical runs with the

HELIX-1 code required 4 hours of CPU time and 4 megawords of memory on the Cray-2 computer.

Significance

All of the crucial rotorcraft performance limitations stem from aerodynamic flow phenomena. The recent success of the British Experimental Rotor Program showed that significant gains in performance can be obtained by designing non-rectangular rotor planforms. Prediction capabilities for transonic effects on drag and pitching moment, and wake-induced blade-vortex interactions, are important tools for future designs of this type. A specific application for this work is the Army's LHX helicopter program, which will put new helicopter designs into the field in approximately 10 years.

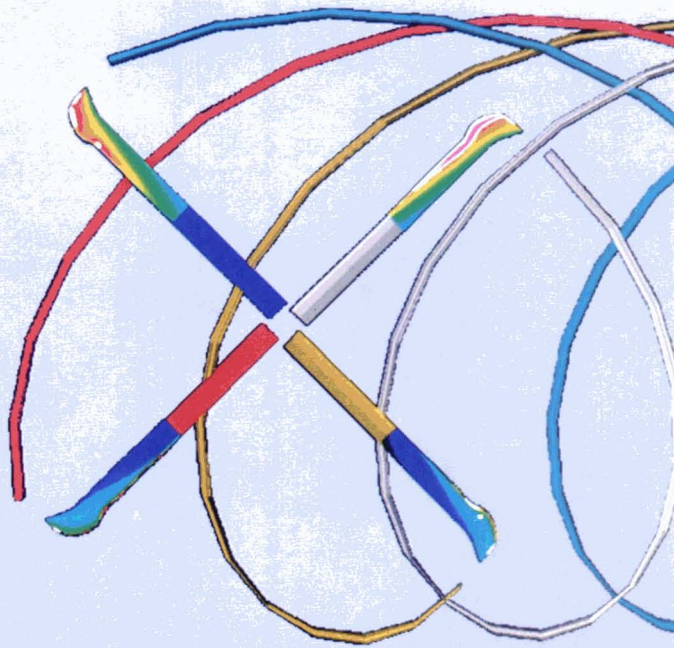
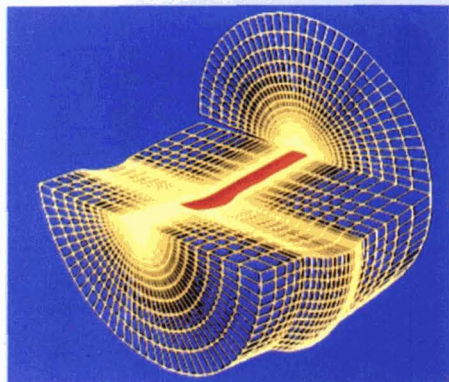
Future Plans

Development of rotor drag and pitching-moment prediction capabilities will continue with an emphasis on forward-flight cases and three-dimensional tip shapes. The Helix-1 rotor-body prediction method will be extended to more complex geometries.

Publications

1. Bridgeman, J. O.; Strawn, R. C.; Caradonna, F. X.; and Chen, C. S. "Advanced Rotor Computations with a Corrected Potential Method." Presented at the 45th Annual Forum of the American Helicopter Society, Boston, MA, May 1989.
2. Strawn, R. C.; Desopper, A.; Miller, J.; and Jones, A. "Correlation of Puma Airloads - Evaluation of CFD Prediction Methods." Presented at the 15th European Rotorcraft Forum, Amsterdam, The Netherlands, Sept. 1989. (See also NASA TM-102226, Aug. 1989.)

— PUMA SWEPT-TIP CASE 3
— $M_{tip}=0.62$, Advance ratio=0.38
— CAMRAD/JA PRESCRIBED WAKE
— FPR SURFACE MACH CONTOURS



computations by Roger Strawn, US Army AFDD

Numerical Simulation of Wave Interactions Related to Transition

Craig L. Streett, Principal Investigator
Co-investigator: S. Balachandar
NASA Langley Research Center

Research Objective

To clarify the role of instability-wave interactions in the middle to late stages of transition in three-dimensional boundary layers.

Approach

The similarity solution for the boundary-layer flow on a rotating disk is used as a prototype for three-dimensional boundary layers that exhibit inflection instabilities known as crossflow modes. These low-frequency disturbances are observed on swept wings as vortices nearly aligned with the potential-flow direction, and are a mechanism for transition in favorable pressure gradient regions on such wings. The rotating-disk model flow has been investigated through hot-wire and flow-visualization experiments, and using linear stability theory. It has been conjectured in the literature that the breakdown of these linear modes is the result of nonlinear interaction of these linear modes; in this study, this interaction is to be elucidated via direct simulation and weakly nonlinear theory.

Accomplishment Description

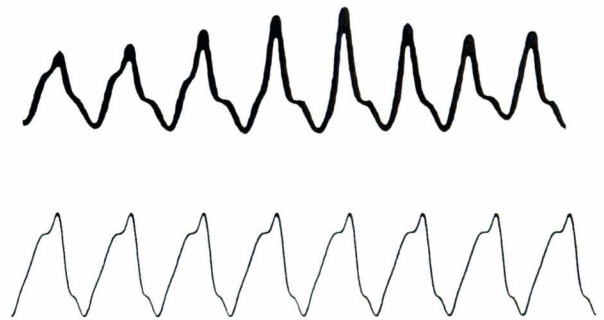
A spectral temporal direct simulation code, written for a preliminary investigation by Malik and Zang, was extensively modified to improve efficiency; a factor-of-seven speedup was achieved. Simulations were carried out using a distribution of primary linear disturbances as initial conditions; the expected nonlinear interactions were not observed. Secondary parametric instability analysis revealed a robust, broadband, Floquet-type instability, appearing at a primary disturbance magnitude threshold of about 4%. The accompanying figure shows a comparison of experimental and theoretical time histories and flow-field visualizations. A new simulation code was written for further improved efficiency; another factor-of-five speedup was achieved. The current code runs at about 40 $\mu\text{sec/pt/set}$ on a $64 \times 64 \times 65$ mesh, at a maximum Courant number of about 0.8. Simulations including energy in the spectral region of the secondary modes confirmed the presence of this instability mechanism; analysis of this well-resolved late-stage simulation is ongoing.

Significance

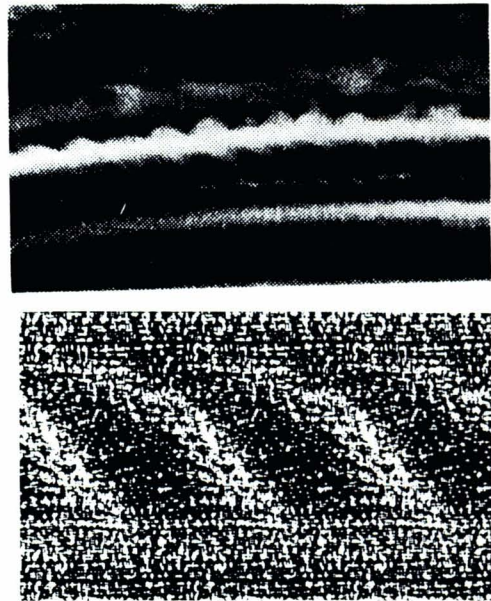
Understanding the stages of breakdown of crossflow disturbances is crucial for developing engineering criteria for transition in flows dominated by these disturbances.

Future Plans

The secondary instability theory is being applied to the nonsimilar three-dimensional boundary layer computed for the swept-wing experiment in the ASU transition wind tunnel; comparisons will be made with detailed hot-wire measurements taken there. Spatial simulations are planned to assess the effects of pressure gradient and nonparallel mean flow on the current (temporal) simulations.



(Top) Hot-wire history (Wilkinson et al.). (Bottom) Streamwise velocity vs. $-\theta$, secondary instability theory.



(Top) Smoke flow visualization (Kohama). (Bottom) Lagrangian particle distribution, secondary instability theory.

Secondary parametric instability of crossflow disturbance, rotating-disk model flow.

Transition Simulations in Nonhomogeneous Geometries

Craig L. Streett, Principal Investigator
NASA Langley Research Center

Research Objective

To develop accurate and efficient methods for the direct simulation of transition in incompressible and compressible flow and in geometries more realistic than hitherto treated; also, to carry out simulations of fundamental transition phenomena.

Approach

Spectral collocation algorithms are developed and utilized for the incompressible and compressible Navier-Stokes equations; nonhomogeneous basis function sets and innovative implementation of boundary treatments are used. The incompressible simulation studies focused on the use of the buffer-domain outflow boundary treatment (developed last operating period) and spectral multidomain methods to improve the accuracy and efficiency of spatial simulations of transition. Multi-tasking techniques were also investigated to improve the efficiency of these simulations. The compressible work involved the development of high-order, fully implicit time-stepping schemes for the compressible Navier-Stokes equations with spectral discretizations.

Accomplishment Description

The spectral, incompressible, spatial simulation code was verified for a number of test cases. These included (1) Poiseuille flow and Blasius boundary layer flow, for which comparisons with linear theory were made; (2) propagation of a vortex in a channel for Poiseuille/Bernard flow in a channel, for which the flow demonstrates a global instability; and (3) interaction of Tollmien-Schlichting waves in a nonparallel boundary layer with a small wall roughness element. Final

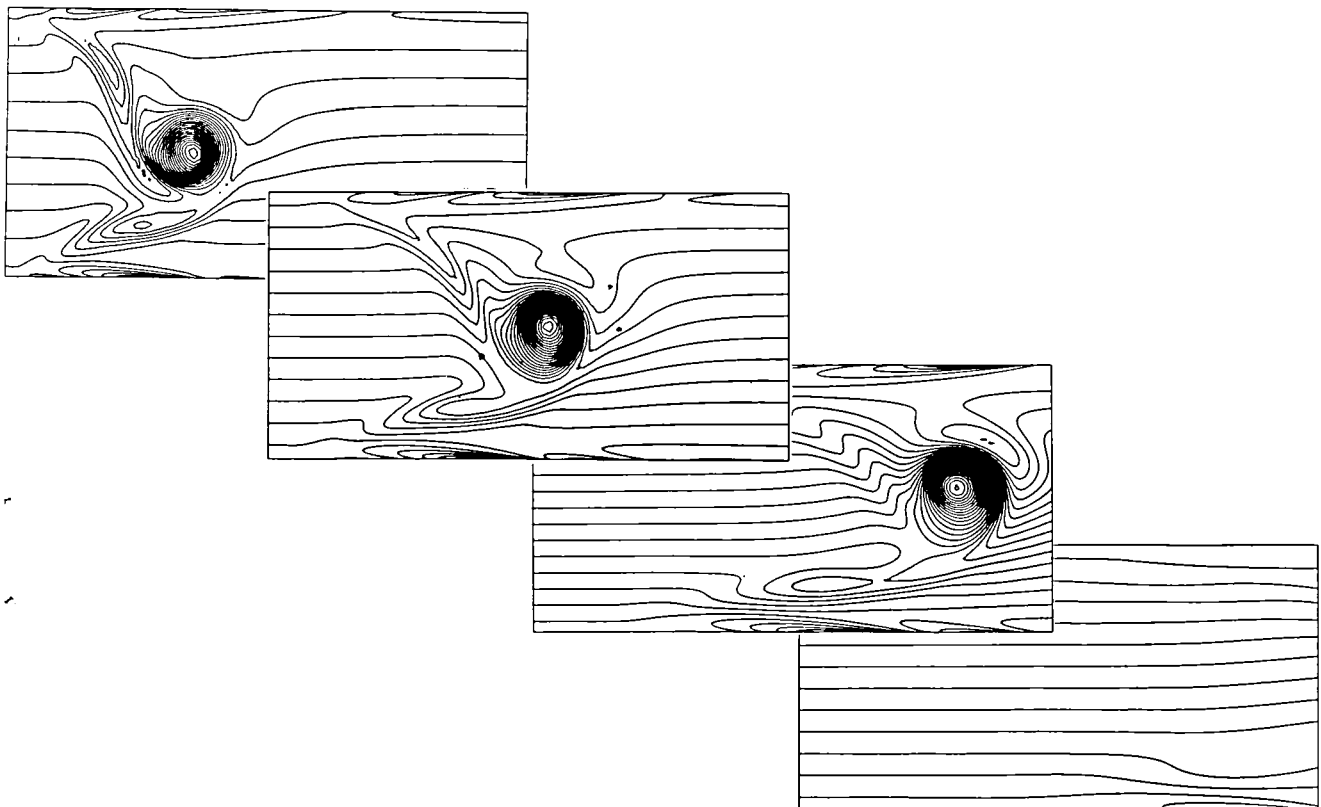
algorithmic refinements are in progress for the case of non-parallel, nonlinear propagation of interacting disturbances in a growing boundary layer. The accompanying figure shows vorticity contours for the vortex-in-channel problem, demonstrating the clear lack of reflection when the vortex passes from the domain. One code developed during this ongoing project was successfully modified to run in a multi-task mode; a speed of 1.6 GFLOPs was achieved on the Cray Y-MP, winning a Cray Gigaflop performance award.

Significance

Virtually all simulations of transition phenomena to date have used the temporal assumption to avoid the requirement of an outflow boundary condition and to simplify the numerics. The present developments permit the use of highly accurate spectral methods for carrying out true spatial simulations of these sensitive flow fields.

Future Plans

Simulations of the effect of pressure gradient and distributed roughness on transition will be carried out in their proper, spatially developing setting. Transition in a separation bubble will also be studied, as will the middle-to-late stages of breakdown of crossflow disturbances. A version of the incompressible spatial code, using high-order finite differences in the streamwise direction, is under development; it will allow the incorporation of geometric complexity. A spectral, fully implicit time-stepping algorithm for the compressible Navier-Stokes equations is being refined, to allow the efficient simulation of compressible boundary-layer and free-shear layer transition.



Incompressible vortex in a channel.

Mars Rover/Sample Return Aerocapture Vehicle

Phil C. Stuart, Principal Investigator
Co-investigator: Chien-Peng Li
NASA Johnson Space Center

Research Objective

The objective is to develop the capability to compute hypersonic flow fields about blunt-nosed bodies traveling at high speeds in a CO_2 atmosphere. This will be used to simulate flows about vehicles performing aerocapture maneuvers at Mars.

Approach

A three-dimensional Navier-Stokes code with ideal-gas and chemical nonequilibrium options is used to predict high-Mach number flows around slender flight vehicles, especially biconics. Initially the Martian atmosphere is modeled as 100% CO_2 dissociating into five species.

Accomplishment Description

An existing, implicit, central-differenced Navier-Stokes code developed at Johnson Space Center was modified to allow the calculation of flows of any dissociating gas. Finite-rate chemistry is modeled by a loosely coupled set of equations. The code has been partially verified for CO_2 . Pressure distributions showed good comparison with data from a low-enthalpy wind tunnel test. Species concentrations at the stagnation point of a sphere showed good agreement with equilibrium values. Calculations were also done on a full-scale Mars aerocapture vehicle. The figure shows a computational grid, pressure distribution, and mass fraction of carbon monoxide on the body surface and in parts of the flow field around a biconic

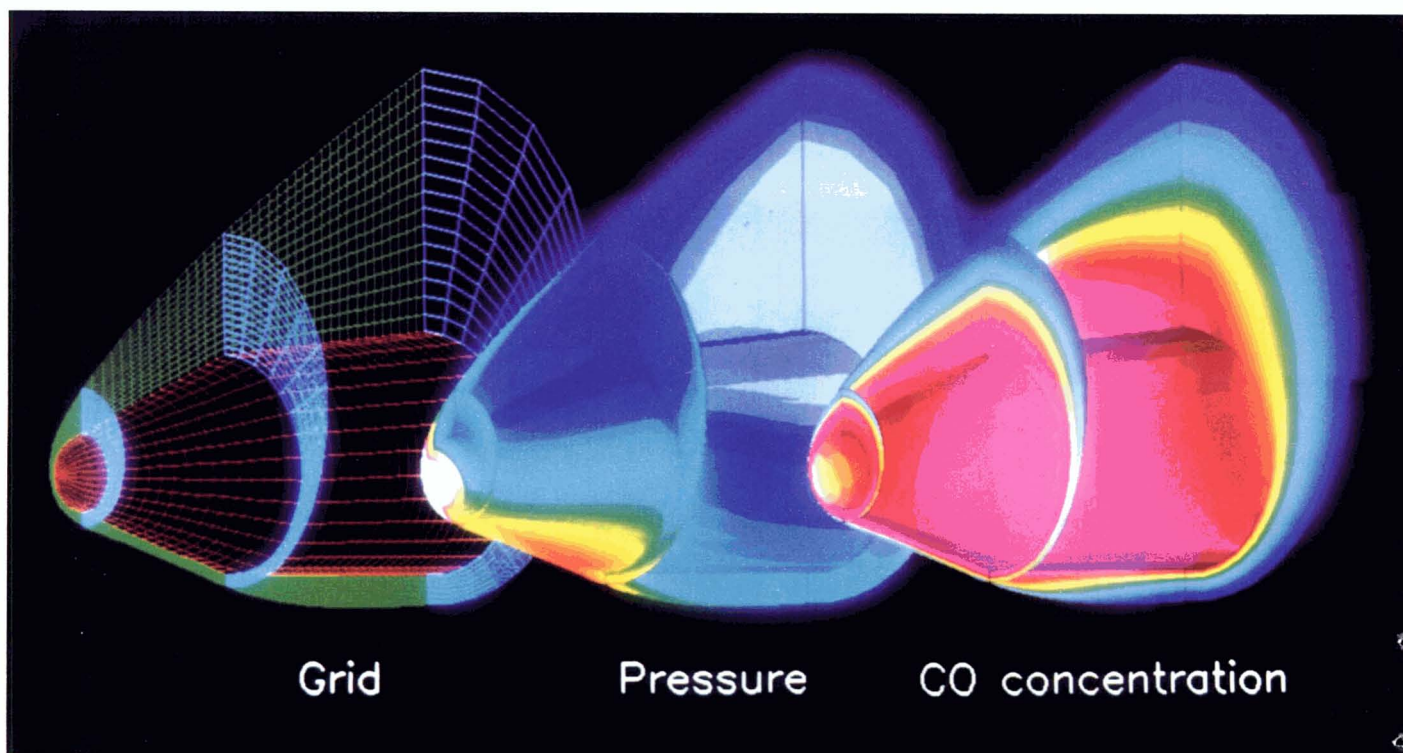
vehicle. The calculation involves free-stream conditions typical of the maximum heating point during an aerocapture maneuver at Mars. The nonequilibrium gas calculations require about 4 Cray-2 hours and 4 megawords of memory when viscosity is not considered.

Significance

Several future missions to Mars are being considered, with at least one in the early design phase. These include both unmanned and manned missions designed to return Martian rock and soil samples to Earth. Because the required landing accuracy is significantly greater than that needed by the Viking landers in 1976, the flight characteristics of the vehicle must be known to a high order of accuracy to ensure mission success. Nonequilibrium CO_2 calculations represent a significant improvement in modeling over what was available in the past.

Future Plans

Our immediate plans are to perform a more comprehensive verification of the code when using CO_2 . The code, with some further modification, will then be used to study the effects of thermal as well as chemical nonequilibrium in a CO_2 atmosphere. In addition, studies of a variety of vehicle shapes will be continued in support of the Mars Rover/Sample Return mission.



Mars Rover/Sample Return aerocapture vehicle in a CO_2 atmosphere.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Numerical Study of Three-Dimensional Viscous Reacting Flows with Application to Scramjet Propulsion

Sundaresa V. Subramanian, Principal Investigator
Co-investigators: Michael J. Epstein
General Electric Aircraft Engines

Research Objective

To optimize and apply existing three-dimensional computer codes for the design and performance improvement of critical hypersonic vehicle components such as inlets, supersonic combustors, and nozzles.

Approach

The three-dimensional, time-dependent, compressible Navier-Stokes equations are solved with an appropriate turbulence model and a physically realistic chemical kinetics model, to describe combustion phenomena occurring in the component of interest.

Accomplishment Description

The computer code RPLUS that was developed at NASA Lewis Research Center was used to successfully predict the complex, three-dimensional, turbulent reacting flows in a supersonic combustor that was also experimentally investigated. Computations were performed for different fuel equivalence ratios (ER) to investigate the extent of fuel penetration, mixing, and reaction. The accompanying figure shows the predicted Mach number distribution at different axial locations

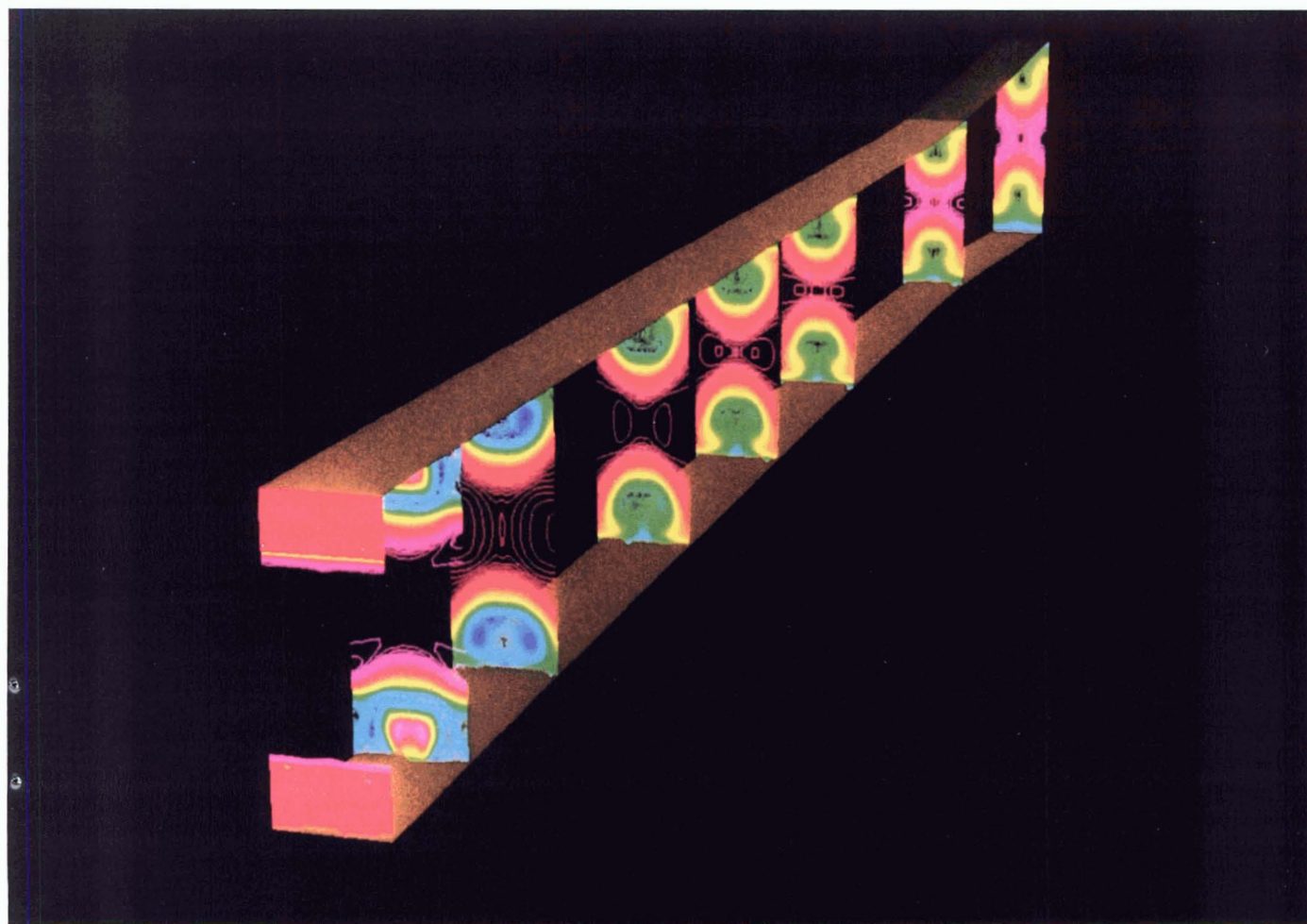
of the combustor for an ER value of 0.76. The fuel is injected normally at three step heights downstream of the step face into the Mach 2.86 free-stream air. In this configuration, the fuel/air interaction produces two distinct counter-rotating vortices that travel the entire length of the combustors, enhancing mixing and reaction. Using an $81 \times 31 \times 21$ grid and nine-species, eighteen-stop, finite-rate chemistry, convergence was obtained after 2600 iterations that required nearly 15 hours and 8 megawords of memory on the Cray Y-MP.

Significance

Enhanced flow prediction methods and codes are beneficial for the design of all air-breathing hypersonic propulsion systems and are essential for the design and development of supersonic combustors that are characterized by a high degree of mixing, reaction, and overall combustion efficiency.

Future Plans

Research is now under way, placing particular emphasis on understanding the mechanisms that are crucial for high-speed mixing and supersonic combustion, such as injector configurations, shock excitation, and turbulence effects.



Mach number distribution at different axial locations of a supersonic combustor; ER = 0.76.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Microwave Hyperthermia Computer Modeling

Dennis Sullivan, Principal Investigator
Co-investigator: Bryan James
Stanford University School of Medicine

Research Objective

The objective is to develop better equipment and methods for deep regional microwave hyperthermia, a relatively new form of cancer therapy that uses microwave heat. Computer simulation is used to design new microwave applicators and to simulate treatments on existing equipment, to increase effectiveness. Specifically, a method has been developed to simulate treatments of patients in an annular phased array, the most commonly used device in the treatment of deep-seated cancer tumors.

Approach

The three-dimensional finite-difference time-domain (FDTD) method is being used to accurately determine the near field of microwave applicators and the body being heated. This method solves the time-dependent Maxwell's equations in difference equation form.

Accomplishment Description

The annular phased array consists of four applicators arranged in an annulus around the patient to be treated. Each applicator is powered by its own linear class A power amplifier, which allows each applicator to have a different amplitude and phase setting. It is this capability that allows the energy to be "steered" within the patient. Using CAT scans, the patient is characterized by the dielectric constants and the conductivities of the various tissue types at 1-cm resolution, creating a model that generally consists of 25,000 to 35,000 cells. The FDTD method is used to determine the energy deposition at every point in the patient model. Such runs require 16 megawords of memory and 500 CPU seconds on the Cray Y-MP. A new technique has been developed to determine the optimum

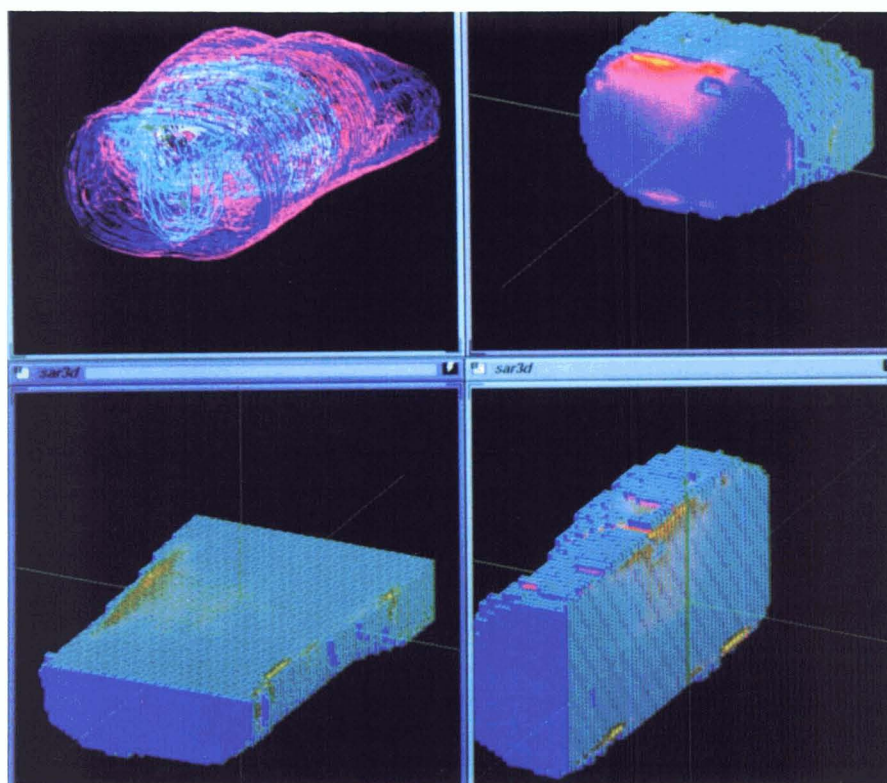
treatment, i.e., maximum tumor heating versus minimum normal-tissue heating. The following procedure is used: With only one quadrant activated, the magnitude and phase of the dominant E-field is saved on disk. This is repeated for all four quadrants at intervals of 10 MHz between 60 and 120 MHz. With this information stored, the magnitude of the E-field at any point in the patient for any combination of amplitude and phase settings of the quadrants at any frequency is determined by the vector sum of the previously stored values, using the principles of superposition and linearity. The resulting energy distribution at the 25,000 to 35,000 points of the patient model is calculated as fast as the operator can type a new set of amplitude and phase settings.

Significance

The problem of treating patients with deep regional hyperthermia has been quantified so that the feasibility of treating different tumor sites can be studied and the optimum phase and amplitude settings can be anticipated.

Future Plans

Determining the best amplitude and phase settings at a given frequency is fundamentally that of optimizing seven independent parameters: four independent amplitude and three independent phase settings (any one phase may arbitrarily be set to zero). With the technique described above, optimization methods may be brought to bear on the problem. However, it is recognized that the annular phased array will always be limited in its ability to localize the energy to the tumor site. For this reason, more effort will be channeled to using the same FDTD to develop a more effective deep-heating applicator.



(Top left) Input to the FDTD program—the model of a patient. Output of the FDTD program—a simulation of the energy distribution of the patient in the annular phased array, showing the transverse (top right), coronal (bottom left), and sagittal (bottom right) views through the body.

Naval Applications of Computational Fluid Dynamics

Chao-Ho Sung, Principal Investigator

Co-investigator: Michael J. Griffin

David Taylor Research Center

Research Objective

The objective is to develop the capability to predict the flow fields about naval ships in order to understand the physics of the flow field. This understanding can then be applied to the innovative design of naval ships.

Approach

The three-dimensional, incompressible, Reynolds-averaged Navier-Stokes equations supplemented by an appropriate turbulence model are to be solved. At present, a scheme based on central-difference, finite-volume, spatial discretization and explicit, one-step, multistage, Runge-Kutta time stepping has been developed. An artificial dissipation model and the Baldwin-Lomax turbulence model are used. Both a local time-stepping and an implicit residual smoothing technique have been used to accelerate the convergence.

Accomplishment Description

Efforts to improve both the accuracy and efficiency of the algorithm continued. During the current operational period, the implementation of a multigrid scheme and the modifications of the artificial dissipation model and the implicit residual smoothing technique were completed. The multigrid scheme increased the convergence rate of three-dimensional turbulent flow computations by three times, at about a 15% increase in

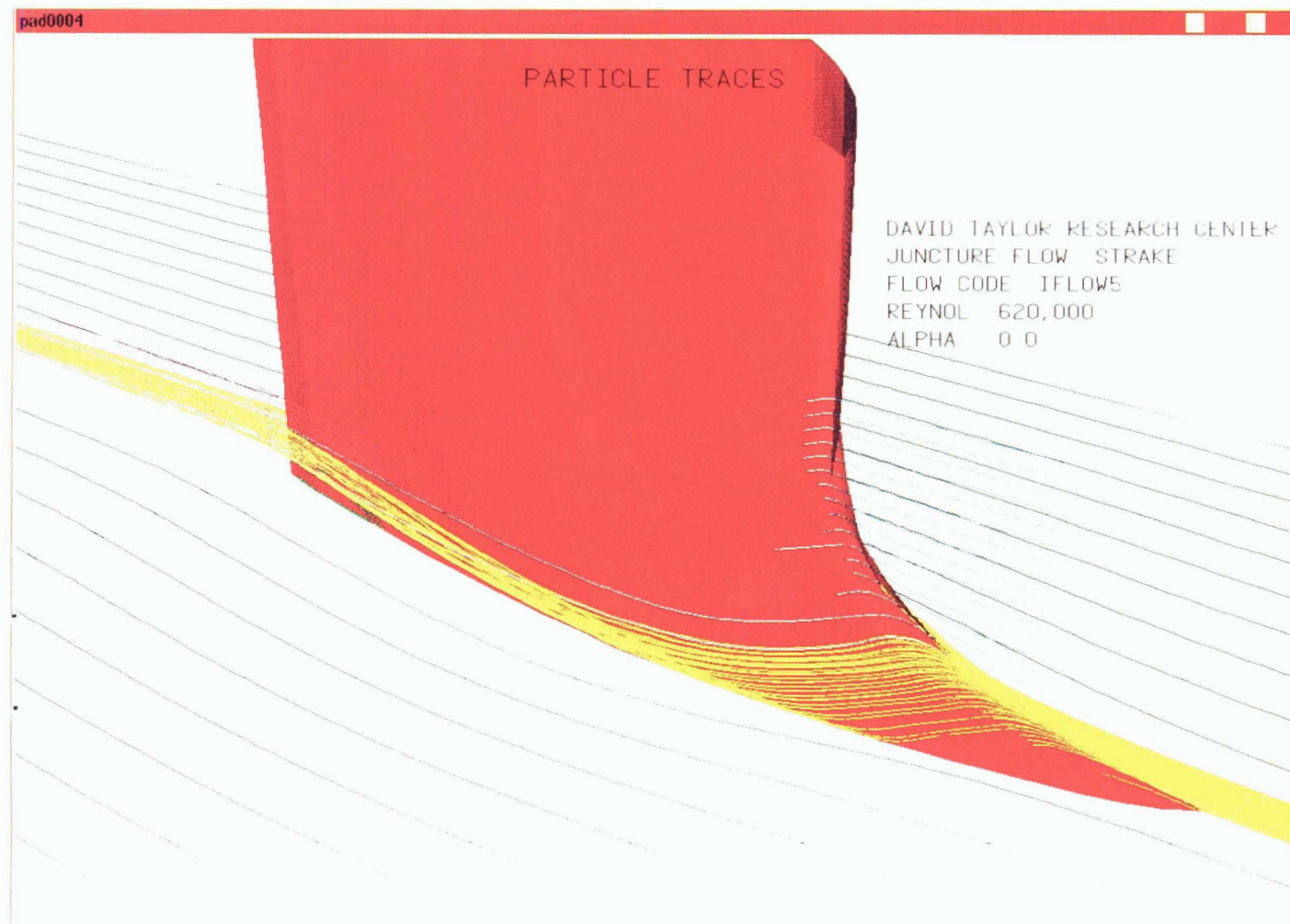
CPU time. The nonisotropic characteristics of the wave propagation were accounted for in the modified artificial dissipation model and in the implicit residual smoothing technique. This modification is essential for very high Reynolds number computations since the aspect ratio is expected to be high. Application to the evaluation of several flow control devices considered for future naval use was also completed. A typical juncture configuration with a $97 \times 25 \times 25$ grid at an angle of attack of 6° requires about 3 megawords and 100 CPU minutes in a single-grid computation for the vorticity to converge.

Significance

The implementation of an efficient multigrid scheme and the development of techniques to improve accuracy for a high-aspect-ratio grid are steps toward large-scale computations with over a million grid points and a very high Reynolds number.

Future Plans

A multizone scheme with emphasis on the stability of abruptly changed grid interfaces will be implemented. Computer codes for the solution of the time-dependent, Reynolds-averaged Navier-Stokes equations will be developed. Both the two-times-scales approach and the projection method will be considered.



Particle traces for strake juncture flow, calculated using the IFLOWS code; $\alpha = 0.0^\circ$, $Re = 6.2 \times 10^5$.

BRUNNEN 1001
COLOR PHOTOGRAPH

Development of Algorithms for Solving the Three-Dimensional Navier-Stokes Equations

R. C. Swanson, Principal Investigator
Co-investigator: M. Sanetrik
NASA Langley Research Center

Research Objective

To develop accurate, efficient, and reliable algorithms for solving the three-dimensional Navier-Stokes equations for all speed regimes.

Approach

Cell-centered and cell-vertex finite-volume spatial discretization (yielding central differencing), a numerical dissipation model, and a multistage time-stepping scheme with acceleration techniques for steady-state calculations are used.

Accomplishment Description

The following elements of the three-dimensional Navier-Stokes solvers have been investigated: (1) variable coefficient implicit residual smoothing, (2) multigrid strategy, and (3) a numerical dissipation model. Transonic flow over an ONERA M6 wing was considered for evaluating the numerics. Alternate forms of residual smoothing were considered, and some improvement in convergence rate was obtained. Modifications in the basic multigrid strategy improved the robustness of the three-dimensional solvers; that is, problems that presented numerical difficulties previously can now be readily solved. A 10 to 15% reduction in computer time was achieved from changes

in these two elements of the schemes. Often, the performance of a scheme is not maintained when fine meshes are used. Therefore, some high-density mesh calculations were performed to confirm the initial evaluation of the improvements to the three-dimensional codes. For the computations, roughly 2 to 50 megawords of storage were required, and the computer time ranged generally from 0.5 to 2.5 hours.

Significance

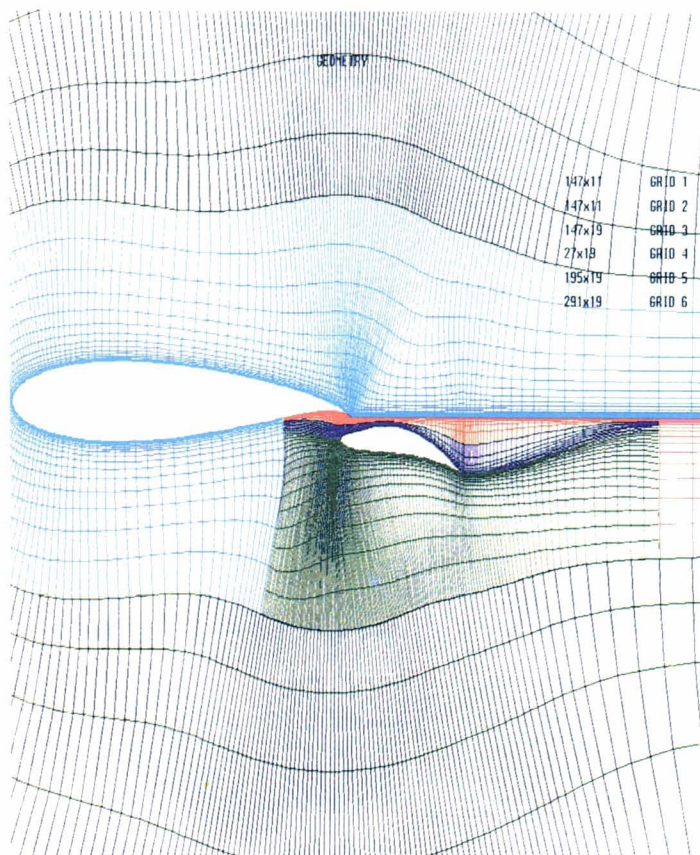
This project has provided significant contributions in the development of the elements currently being used in an efficient generation of Navier-Stokes codes. Much more robust algorithms for three-dimensional viscous flow computations have been constructed.

Future Plans

We plan to develop effective and accurate algorithms for solving high-speed flows, and to obtain complete validation of the domain-partitioning procedure.

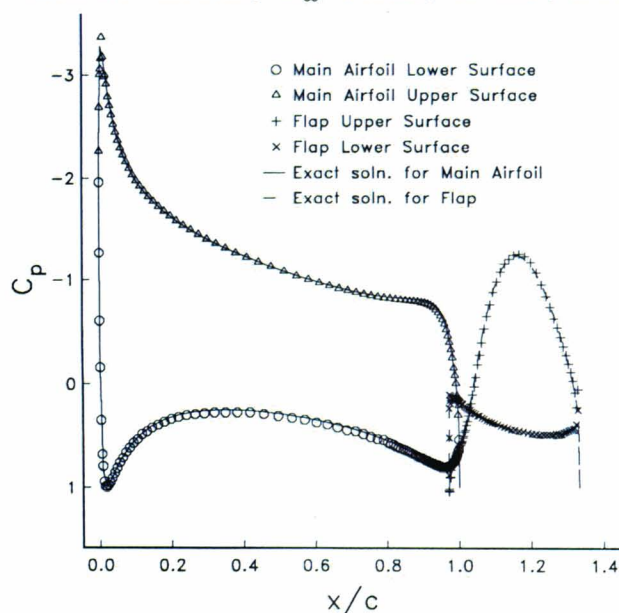
Publications

"An Efficient Cell-Vertex Multigrid Scheme for the Three-Dimensional Navier-Stokes Equations." Presented at the AIAA 9th Computational Fluid Dynamics Conference, Buffalo, NY.



Domain decomposition for a two-element airfoil.

Karman-Trefftz, $M_\infty=0.125$, $\alpha=0.0$, Inviscid



2025 RELEASE
NATIONAL ARCHIVES

ORIGINAL PAGE
COLOR PHOTOGRAPH

Hydrodynamic Prediction Methods for Advanced Submarine Configurations

Tsze C. Tai, Principal Investigator

Co-investigators: Steven C. Fisher, Cheng-Wen Lin, and Gerald D. Smith

David Taylor Research Center

Research Objective

To develop a computational fluid dynamics (CFD) method for the analysis and design of submarine hull/appendage configurations, and to apply the method together with numerical optimization techniques to design novel submarine shapes that are based on specified mission requirements.

Approach

The basic tool is a three-dimensional, full or thin-layer, Reynolds stress-averaged Navier-Stokes solver subject to steady and unsteady free-stream conditions. A research code for incompressible flow was developed to determine the subject submarine flow field. A zonal approach was used whereby the flow field was subdivided into smaller zones. Steady and unsteady flows over a submarine hull/appendage configuration at straight-ahead even keel and also at moderate turning will be considered. The approach will then be extended to high-speed maneuvering conditions with flow separation and free vortex flows.

Accomplishment Description

A multiblock grid-generation code (based on the algebraic method) was developed and used to generate surface and volume grids for submarine geometries. An incompressible, steady Navier-Stokes code with multizonal capability was used to compute the viscous flow around several complex configurations, including a fully appended submarine. In addition, flow computations were conducted to simulate experimental conditions for a submarine model in a wind tunnel. Results of these computations illustrate the effects of the supporting struts and wind tunnel walls on the experimental data. Computed velocity contours for a fully appended submarine model

in a wind tunnel are illustrated in the accompanying figure. This solution required 250,000 grid points, took approximately 2.5 hours of computing time on the Cray-2, and used 20 megawords of memory.

Significance

The ability to predict the flow field around real submarine geometries is an indispensable tool for naval designers. In addition, pretest analysis and planning can be enhanced by applying the developed computational techniques to experimental conditions. The understanding of the physical phenomena on novel hull shapes or control surfaces can be enhanced and numerical computational results can be obtained with a reasonable turnaround time.

Future Plans

A more interactive grid generator will be used and the flow solver will be further improved in numerical accuracy and robustness. The flow solver will also be made more interactive and user-friendly. In addition, computational speed and memory usage are targeted for improvement so that a more practical computational design tool can be achieved.

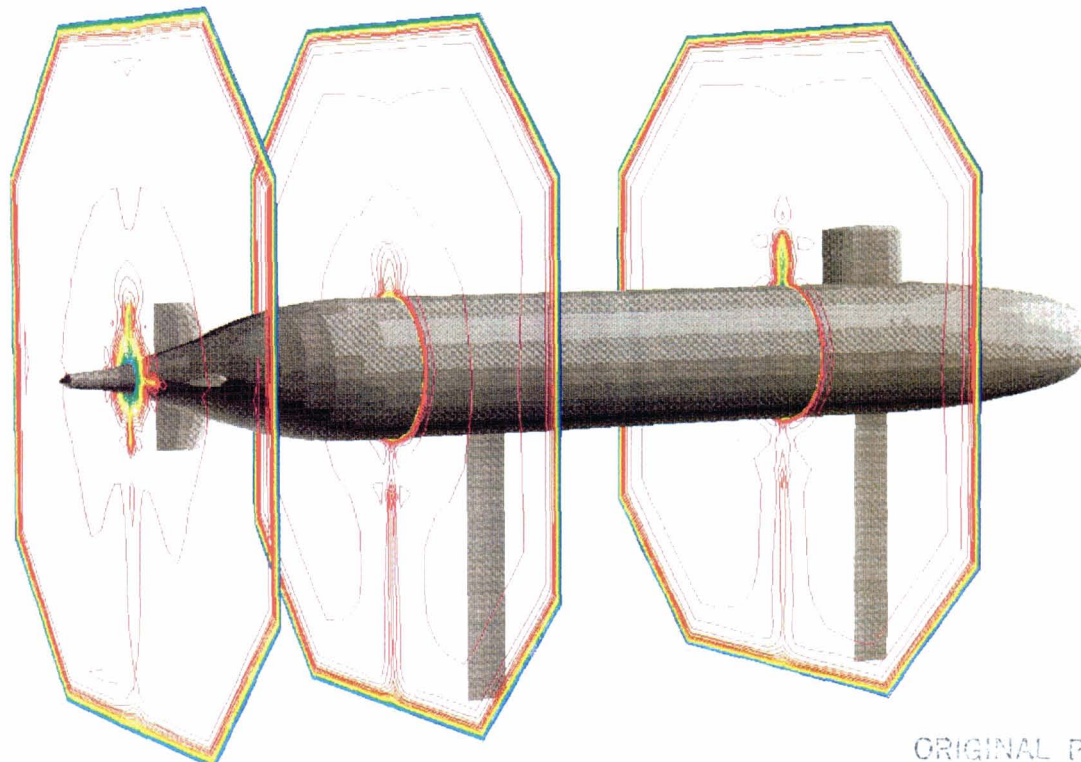
Publications

1. Lin, Cheng-Wen; Smith, Gerald D.; and Fisher, Steven C. "Numerical Flow Simulations on the DARPA SUBOFF Configurations." David Taylor Research Center/Ship Hydromechanics Department Report, Apr. 1990.
2. Lin, Cheng-Wen; Smith, Gerald D.; and Fisher, Steven C. "Validation of Numerical Wind Tunnel Flow Simulation on the DARPA Bare Body." David Taylor Research Center/Ship Hydromechanics Department Report, Apr. 1990.

U Velocity

Contour Levels

-0.00000
0.00000
0.04000
0.08000
0.12000
0.16000
0.20000
0.24000
0.28000
0.32000
0.36000
0.40000
0.44000
0.48000
0.52000
0.56000
0.60000
0.64000
0.68000
0.72000
0.76000
0.80000
0.84000
0.88000
0.92000
0.96000
1.00000
1.04000
1.08000
1.12000



ORIGINAL PAGE
COLOR PHOTOGRAPH

Computed velocity contours for a submarine model in a wind tunnel.

Low-Speed Maneuver Aerodynamics in a Nonuniform Free Stream

Tsze C. Tai, Principal Investigator
David Taylor Research Center

Research Objective

The objective is to develop a computational method for determining the nonlinear aerodynamics of an aircraft/ship interface environment. Of particular interest to the Navy is the ability to analyze the effects of the airwake, crosswind, wind-speed fluctuation, and gust on the lift characteristics of an aircraft during a landing maneuver. Steady and unsteady flow conditions will be considered.

Approach

The three-dimensional, thin-layer Reynolds stress-averaged Navier-Stokes equations will be solved subject to a low-speed nonuniform free stream. A zonal approach will be adopted whereby the flow field is subdivided into smaller zones. This approach allows suitable grids to be generated for different zones with moderate memory requirements. A fast-convergent diagonal algorithm of the Beam-Warming type will be used to solve the discretized equations. The NASA Ames ARC3D code and its zonal-version transonic and compressible Navier-Stokes (TNS and CNS) codes will be used as the basic program, with necessary modifications. Steady flows, with a nonuniform free stream, of a ship wake or a crosswind will be considered first. The approach will be extended to unsteady conditions by considering wind-speed fluctuations and/or a sudden gust.

Accomplishment Description

Computational results for flow over an F-14A wing based on a thin-layer Navier-Stokes method are examined, and resulting flow separation patterns investigated. The wing is fixed with a sweep of 20° and travels at an altitude of 45,000 ft above sea level at Mach 0.6, and at sea level at Mach 0.1, at various angles of attack. These conditions, which yield a Reynolds number of 8.95×10^6 , allow evaluation of the effect of Mach number on flow separation patterns with fixed Reynolds

number. Massive flow separation occurs at Mach 0.6 at an angle of attack of 10° . At Mach 0.1, with the same Reynolds number, the rear-region massive separation is replaced by a moderate leading-edge separation. Consequently, the lift and drag values are far more stable as the angle of attack increases. A typical case that used a four-zone grid for the flow over a wing took approximately 2 hours of CPU time on the Cray-2 with 10 to 15 megawords of memory. These results, mainly based on the Ames TNS code with some modifications for low-speed flow conditions, are summarized in AIAA Paper 90-0596. The nonuniform stream effect was evaluated by imposing an analytical defective function representing the free stream. A small decrease in lift resulted. An overall grid for the F-14A wing-fuselage-tail configuration was generated using the Ames 3DGRAPE code; it is shown in the accompanying figure. Implementation of zonal techniques based on the Ames CNS code is in progress.

Significance

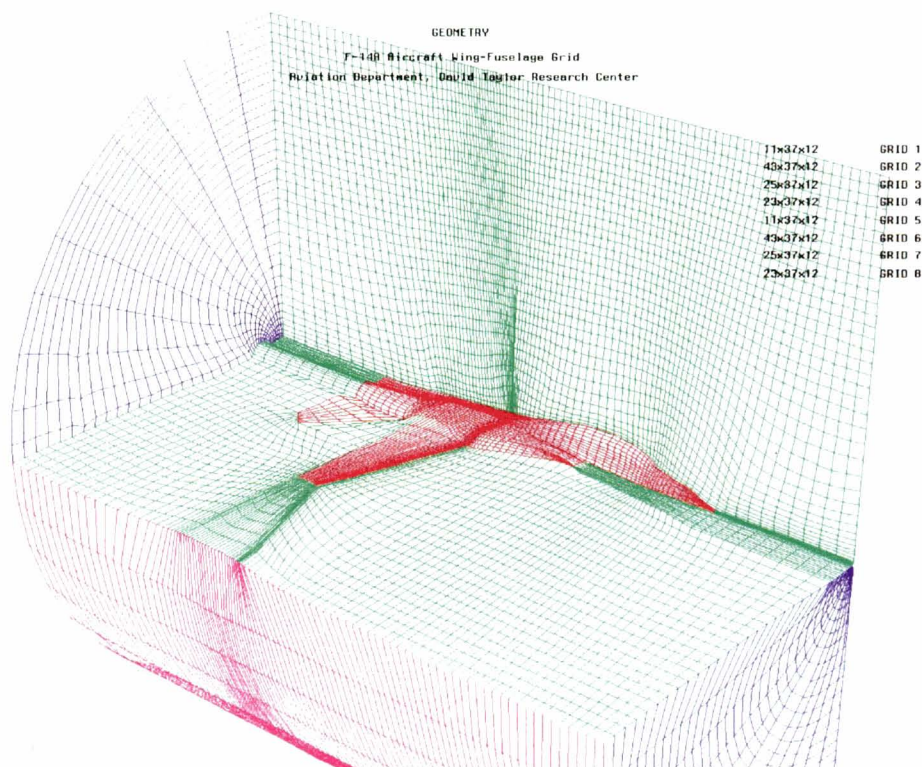
A need for enhanced fighter maneuver (EFM) has been identified in the Naval Aviation Plan. Although the EFM concept is applicable to any fighter aircraft, the Navy operation is unique in that it requires aircraft to be able to maneuver under nonuniform free-stream conditions, such as a crosswind or a gust near a ship deck. In developing the EFM technology, this research will focus on problems specific to naval aviation.

Future Plans

The Ames CNS and 3DGRAPE codes will be modified to handle a complete F-14A aircraft solution. Steady flow with uniform and nonuniform free-stream conditions will be considered in the NAS 1990-91 operational year.

Publications

Tai, T. C. "Flow Separation Patterns Over an F-14A Aircraft Wing." AIAA Paper 90-0596, Jan. 1990.



Grid geometry for an F-14A wing-fuselage-tail configuration.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Unsteady Euler Analysis of the Redistribution of an Inlet Temperature Distortion in a Turbine

Ronald K. Takahashi, Principal Investigator

Co-investigator: Ron-Ho Ni

United Technologies, Pratt & Whitney

Research Objective

To analyze the effects of three-dimensional inflow temperature and pressure distortions on the local gas temperatures in rotor passages of turbine stages.

Approach

A time-accurate three-dimensional Euler code with surface shear modeling was used to simulate a low-speed experimental turbine rig. An inlet high-temperature "jet" was modeled to simulate turbine-engine combustor exit conditions.

Accomplishment Description

The results showed dramatic effects of secondary flow on three-dimensional temperature migration, the process by which the hot combustor gas redistributes within the rotor passage. The accompanying figure is an instantaneous image of the hot gas as it impacts the rotor pressure side, where it lingers and spreads out toward the end walls because of secondary flow. The calculations match the heating patterns observed both in experiments and on actual engine parts, especially on the rotor suction and pressure sides, the platform, and the outer air seals. This good agreement gives confidence that even the coarse mesh is resolving at least the first-order physics controlling the temperature migration process. Calculations using a finer computational grid showed

that the temperature migration process is very sensitive to grid density in the range of the present grid sizes, and further work is required to quantify the effects of grid size. Coarse grid calculations require about 7 Cray-2 hours per run, and the fine grid requires about 25 Cray-2 hours and 8 megawords of memory.

Significance

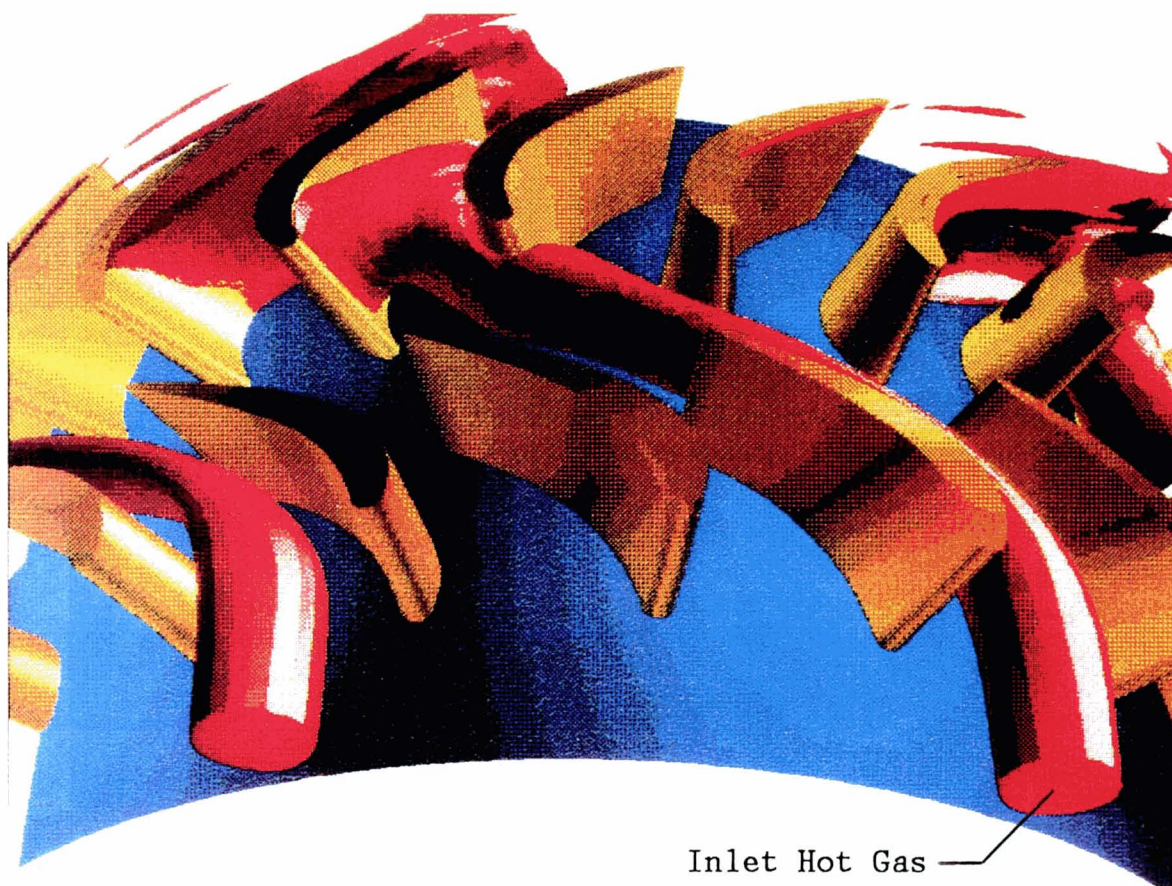
The redistribution of inlet temperature distortions in the rotor passages is believed to be a major cause of localized overheating of rotor airfoils and passage end walls. The computations have added to the understanding of the physics controlling the temperature migration process, and the unsteady code may prove to be a useful design tool for increasing the lives of turbine parts.

Future Plans

Continuation of this work in 1990 will focus on grid sensitivity and the effects of tip leakage on temperature migration.

Publications

Takahashi, R. K., and Ni, R. H. "Unsteady Euler Analysis of the Redistribution of an Inlet Temperature Distortion in a Turbine." Presented at the AIAA/ASME/SAE/ASEE 26th Joint Propulsion Conference, Orlando, FL, July 1990.



Rotors

Stators

Inlet Hot Gas

ORIGINAL PAGE
COLOR PHOTOGRAPH

Instantaneous image of an isotherm contour illustrating the interaction of the hot gas with the passing rotors.

Thermochemical and Radiative Nonequilibrium Flow Simulation for the Aeroassist Flight Experiment

Luen T. Tam, Principal Investigator

Co-investigator: Chien P. Li

Lockheed Engineering and Sciences Company/NASA Johnson Space Center

Research Objective

To develop and validate a three-dimensional thermochemical and radiative nonequilibrium model for the inviscid/viscous flow simulation of the Aeroassist Flight Experiment (AFE) vehicle.

Approach

Two- and three-dimensional flow fields were obtained from the Navier-Stokes equations coupled with conservation of chemical species, overall energy, electron translational energy, and molecular-vibrational energy. The method includes generating grids that are adaptive to the shock and body, and controlling numerical dissipation by using local flow gradients and total enthalpy.

Accomplishment Description

A two-temperature thermal nonequilibrium model was developed and incorporated into the viscous reactive flow (VRFLO) code. Preliminary results indicated that strong dissociation and weak ionization take place at the forebody and that vibrational freezing occurs in the afterbody expansion region, where the vibrational temperature is even higher than the translational temperature. The accompanying figure shows the electron number density contours obtained from two thermal models at the AFE trajectory point corresponding to the maximum aerodynamic heating. This comparison suggests that a 7-species kinetic model fails to predict the electron

number density accurately. Typically, 20 Cray-2 CPU hours were required for a three-dimensional AFE complete-body viscous-flow solution with 2000 iterations, calculated from a two-temperature, 11-species, 30-reaction model on a grid of $30 \times 60 \times 19$. The memory requirement was close to 3.8 megawords.

Significance

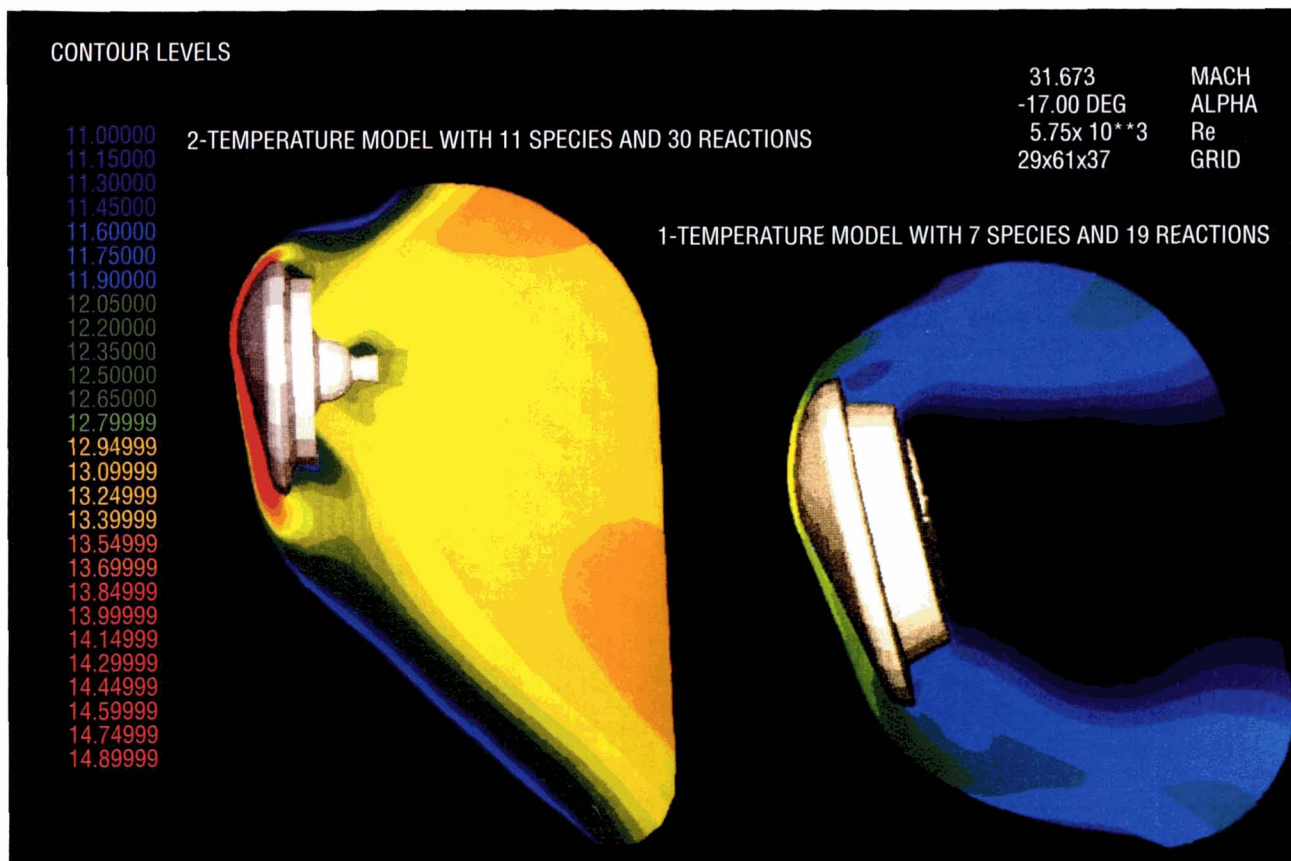
The AFE flight data related to surface properties, heating, electron number density, and AFE wake characteristics will be used to validate the VRFLO code.

Future Plans

Although the two-temperature model is being used to assess the AFE complete-body viscous flows, a radiation model also needs to be incorporated into the study since the radiative heat transfer from the forebody could be of importance as well as that from the wake.

Publications

1. Tam, L. T., and Li, C. P. "Three-Dimensional Thermochemical Nonequilibrium Flow Modeling for Hypersonic Flows." AIAA Paper 89-1860, June 1989.
2. Tam, L. T., and Li, C. P. "Comparisons of Thermochemical Nonequilibrium Viscous Flowfield Predictions for AFE Vehicle." AIAA Paper 90-0141, Jan. 1990.



Electron number density contours, $A \cdot \log_{10}(\text{electron number density})/\text{CM}^3$; $M = 31.673$, $\alpha = -17.00^\circ$, $Re = 5.75 \times 10^3$, grid size $29 \times 61 \times 37$. (Left) Two-temperature model with 11 species and 30 reactions. (Right) One-temperature model with 7 species and 19 reactions.

Development of a Robust Parabolized Navier-Stokes Code for Computing Three-Dimensional, Chemically Reacting Flow Fields

John C. Tannehill, Principal Investigator

Co-investigators: Phil E. Buelow and John O. Ivalts

Iowa State University

Research Objective

To develop a robust computer code to compute the three-dimensional, viscous, chemically reacting flow around hypersonic vehicles.

Approach

The three-dimensional parabolized Navier-Stokes (PNS) equations for chemically reacting flows are solved using an upwind total-variation-diminishing (TVD) finite-difference scheme.

Accomplishment Description

A new upwind PNS code was developed to compute the three-dimensional flow of chemically reacting air around hypersonic vehicles. The code is a modification of the perfect-gas, three-dimensional UPS code of Lawrence et al., which was extended in the current study to permit the calculation of hypersonic, viscous flows in chemical nonequilibrium. The algorithm solves the PNS equations using a finite-volume, upwind TVD method based on Roe's approximate Riemann solver, which was modified to account for real-gas effects. The current code solves the fluid dynamic and species continuity equations in a loosely coupled manner. The fluid medium is assumed to be a chemically reacting mixture of thermally perfect (but calorically imperfect) gases in thermal equilibrium.

If desired, perfect-gas or equilibrium-air calculations can also be performed with the same code. Results were obtained for the hypersonic laminar flow over cones at various angles of attack and over the McDonnell Douglas Generic Option hypersonic vehicle.

Significance

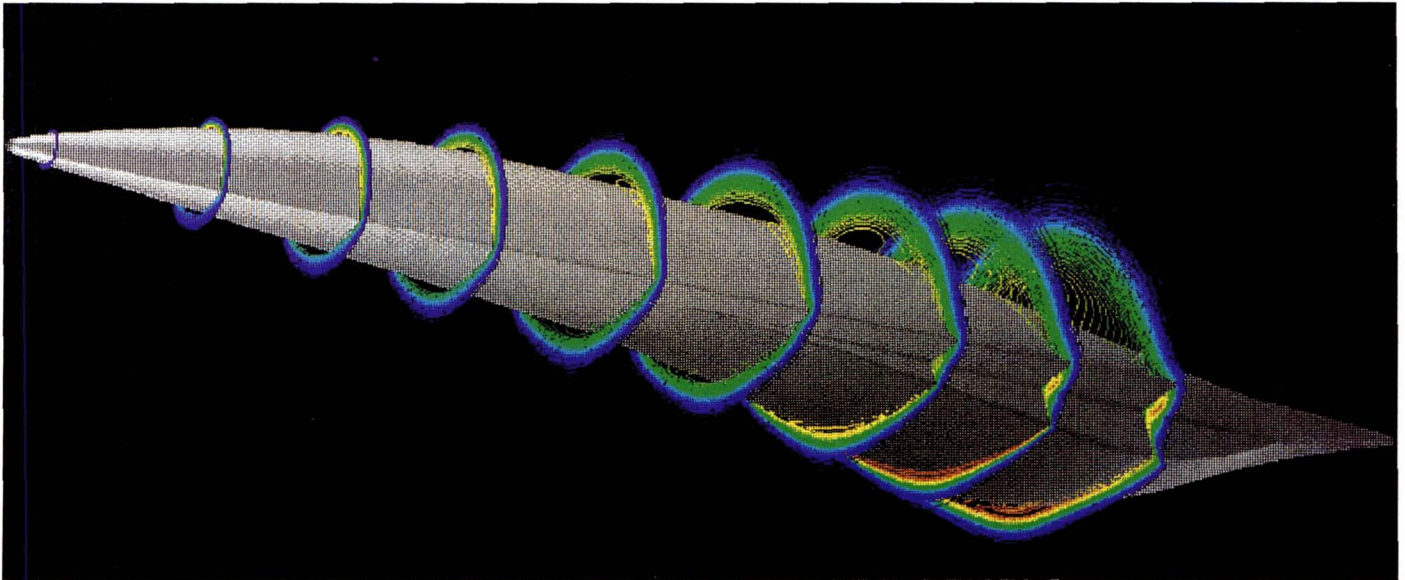
This code provides the ability to compute the chemically reacting flow field about the National Aero-Space Plane.

Future Plans

The code is currently being extended to permit the calculation of internal flows with hydrogen-air chemistry. The code will then be able to compute aerodynamic and propulsive flow fields simultaneously.

Publications

1. Tannehill, J. C.; Ivalts, J. O.; Buelow, P. E.; Prabhu, D. K.; and Lawrence, S. L. "Upwind Parabolized Navier-Stokes Code for Chemically Reacting Flows." *J. Thermophysics and Heat Transfer* 4, no. 2 (Apr. 1990): 149-156.
2. Tannehill, J. C.; Buelow, P. E.; Ivalts, J. O.; and Lawrence, S. L. "Three-Dimensional Upwind Parabolized Navier-Stokes Code for Real Gas Flows." *J. Spacecraft and Rockets* 27, no. 2 (Mar.-Apr. 1990): 150-159.



Computed atomic oxygen levels for the McDonnell Douglas Generic Option II BWB Aerothermal Model; a PNS solution with finite-rate air chemistry: $M_\infty = 25.3$, $\alpha = 0^\circ$, altitude = 61 km.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Computation of Three-Dimensional Flows Using Unstructured Grids

Rajiv Thareja, Principal Investigator

Co-investigators: Ken Morgan, Jaime Peraire, Obey Hassan, and Joaquin Peiro

NASA Langley Research Center

Research Objective

To develop a general method for automatically discretizing into unstructured assemblies of tetrahedra the three-dimensional solution domain of complex shapes, and to obtain a solution to transonic and hypersonic flows over realistic configurations.

Approach

An advancing front mesh generator was coupled with a higher-order-accurate Taylor-Galerkin flow solver to compute flows over arbitrary complex geometries. Considerable benefits were achieved because the mesh was adapted in an optimal manner. The method was used to benchmark a solution for transonic flow at a Mach number of 0.84 and 3.06° angle of attack over an ONERA M6 wing against several other solution methods in a workshop held at NASA Langley Research Center in January 1990. The computation was performed on a grid provided for the participants and consisted of 231,507 tetrahedral elements and 42,410 points. Views of the discretization of the upper and lower surfaces of the wing, along with the computed pressure contours, are shown in the figure. Even though the mesh is relatively coarse, a lambda shock is captured on the upper surface. The adaptive feature of the scheme is also demonstrated in the computation of internal flow in an engine inlet with a free-stream Mach number of 6. The computational domain is a region bounded by planar compression surfaces along three sides. The initial mesh consists of 11,478 triangular surface elements, 155,702 tetrahedra, and 29,531 mesh points. Mach number contours on this mesh show the complex shock interactions and reflections. Using this solution and an error indicator that uses the

second derivatives of density and Mach number, a second mesh consisting of 11,290 triangular surface elements, 181,292 tetrahedra, and 33,786 points was generated as shown, along with the computed Mach number contours. The improvement in the definition of the flow features on the adapted mesh is apparent. It should be noted that the resolution on this mesh was increased by a factor of 2.5 although the number of mesh points increased only slightly.

Accomplishment Description

An automatic procedure that combines adaptive mesh generation using unstructured grids with a flow solution for complex geometries was benchmarked against several competing solution methods, with very favorable agreement.

Significance

The method can be used to model hypersonic flows over arbitrary, complex, three-dimensional geometries. Very good performance in terms of accuracy and efficiency has been achieved. Mesh resolution can be achieved with only a minimal increase in the number of points in the computational domain.

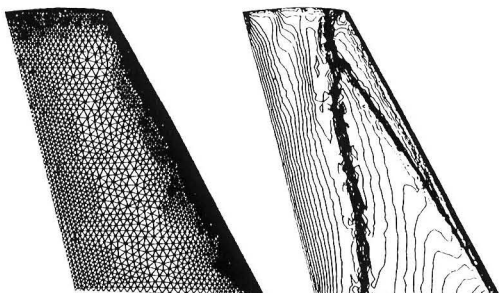
Future Plans

Future plans include developing higher-order-accurate finite-volume algorithms and higher vectorization of these codes.

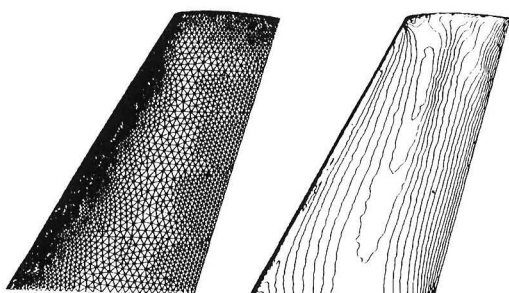
Publications

Morgan, K.; Peraire, J.; Peiro, J.; and Hassan, O. "The Computation of Three Dimensional Flows Using Unstructured Grids." Presented at the Computational Fluid Dynamics Conference, Minneapolis, Apr. 1990.

UPPER SURFACE

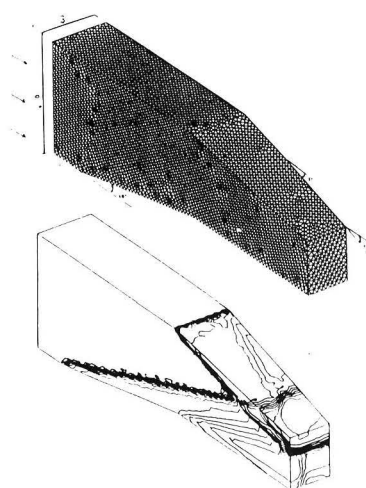


LOWER SURFACE

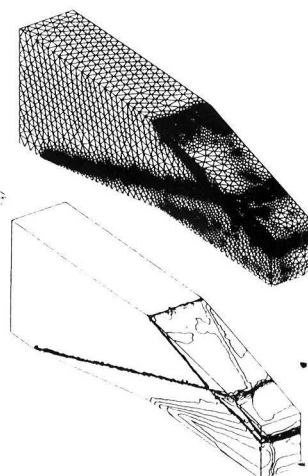


Meshes and pressure contours for transonic flow over an ONERA M6 wing.

INITIAL MESH



ADAPTED MESH



Meshes and Mach number contours for hypersonic flow in an engine inlet.

Viscous Hypersonic Flow over a 24-Degree Compression Corner

Rajiv Thareja, Principal Investigator
Co-investigators: Ken Morgan and Jaime Peraire
NASA Langley Research Center

Research Objective

To study the effect of mesh adaptivity on the solution of Mach 14.1 viscous flow over a 24° compression corner, and to compare the solutions from a higher order, unstructured-grid upwind code with other numerical results and with experiment.

Approach

A higher order, adaptive unstructured-mesh generator and flow solver called LARCNESS was developed and used to compute viscous Mach 14.1 flow over a 24° compression corner with a Reynolds number of 72,000/ft. The solutions show good overall agreement with CFL3D solutions on a structured 101 × 101 mesh (denoted as mesh SM), but the calculations do not compare well with experiment. An earlier study had concluded that the flow has significant three-dimensional effects, and a two-dimensional solution is not able to provide agreement with experiment. However, refining the leading-edge mesh while retaining the structure of the mesh by moving the computational domain closer to the wall (denoted as mesh MSM) leads to significantly improved comparison with experiment. Very good agreement with CFL3D results was also observed for this mesh. Because of the refined mesh, the leading-edge shock moves closer to the body, resulting in an impingement location significantly upstream of the SM solution. An adaptively generated mesh aligns the mesh with the leading-edge shock and the reflected shock, resulting in improved shock definition and a slight improvement in the predictions of flow separation and reattachment, as compared with experiment and CFL3D flow solutions. The computed v-velocity contours for the three meshes show the effect of the mesh in improving the flow features. Although the adapted mesh has about the same

number of nodes as the structured meshes, it has finer mesh resolution where required, resulting in an optimal number of nodes.

Accomplishment Description

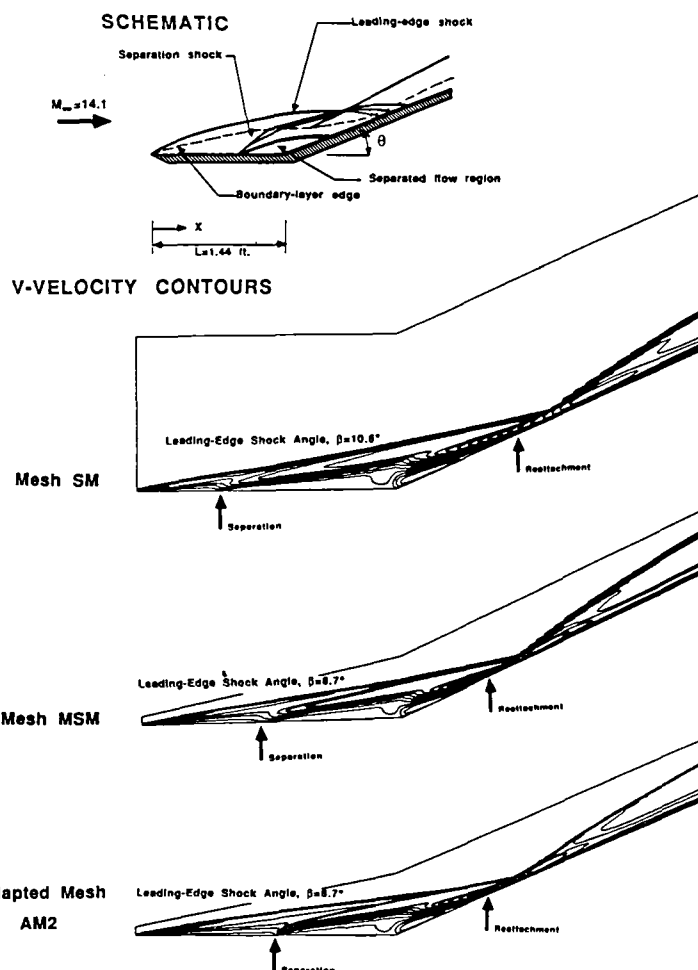
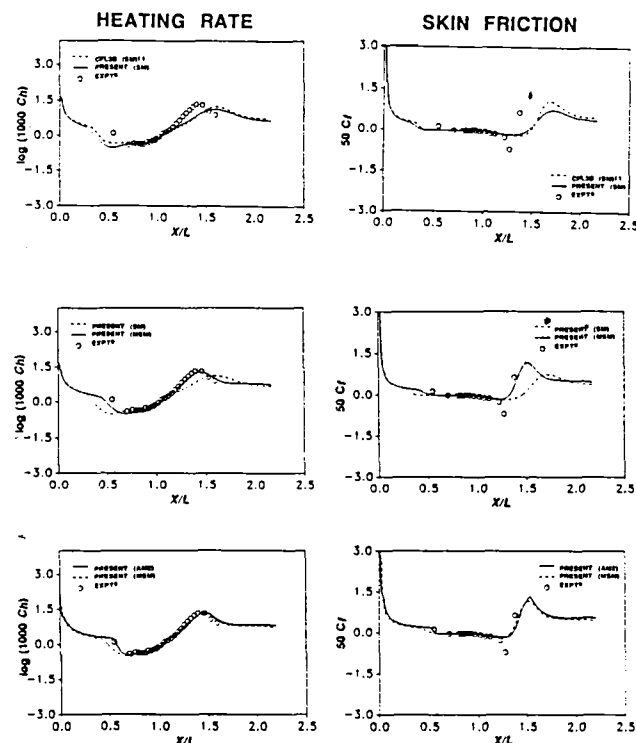
The effect of mesh size near the leading edge of a compression corner was shown to significantly influence the downstream features. An automatic procedure that combines adaptive mesh generation using unstructured grids with a flow solution for complex geometries was benchmarked against several competing solution methods, with very favorable agreement.

Significance

The method shows that mesh resolution at the leading edge has a significant impact on the location of the leading-edge shock and the resulting locations of peak pressure and heating rates, and on the location of separation and reattachment points. Mesh adaptivity can be used to place elements aligned with the principal features of the flow. Substantial mesh resolution can be achieved with only a minimal increase in the number of points in the computational domain.

Future Plans

Future plans include computing an adapted unstructured-grid solution over the three-dimensional model of the compression corner.



A schematic of the flow, comparisons of computed and experimental heating rate and skin friction, and v-velocity contours for three different mesh types.

Scattering Theory and Calculations for Chemical Reactions and Molecular Energy Transfer

Donald G. Truhlar, Principal Investigator

Co-investigators: David C. Chatfield, Philippe Halvick, David W. Schwenke, Thanh N. Truong, Michael J. Unekis, and Meishan Zhao

University of Minnesota

Research Objective

The objectives of this work are to develop and implement efficient and accurate methods for calculating the quantum mechanical dynamics of reactive and inelastic collisions of atoms and molecules, and to use them to carry out benchmark calculations that can be compared with experiment. Both atom-diatom reactions and diatom-diatom energy transfer collisions are under study.

Approach

The quantum mechanical equations governing the collision processes are solved by algebraic variational methods and propagation techniques. All calculations are highly vectorized.

Accomplishment Description

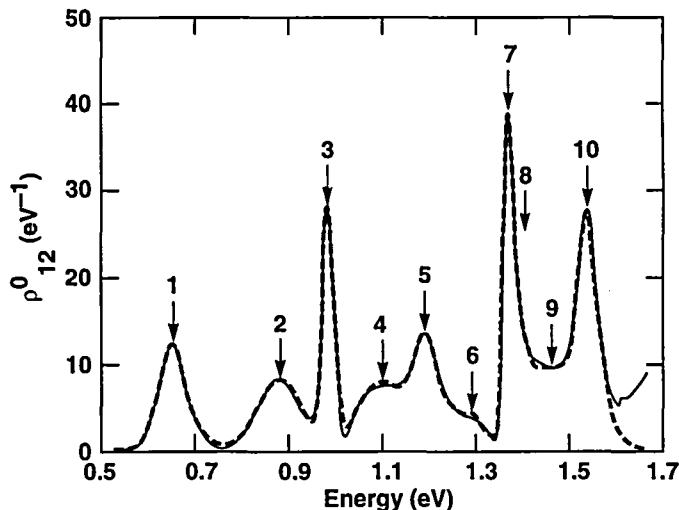
We have developed very efficient codes, as described above. In one study of reactive collisions, we obtained evidence that the accurate quantum mechanical probability of the reaction of H with H_2 is globally controlled by quantized transition states up to very high energies. The quantized transition states produce steplike features in the cumulative reaction probability curves that are analyzed, up to energies of 1.6 eV; the analysis clearly associates these steps (or "thresholds") with quantized dynamical bottlenecks that control the passage of reactive flux to products. We have assigned bend and stretch quantum numbers to the modes orthogonal to the reaction coordinate for all these transition states, based on threshold energies of semiclassical, vibrationally adiabatic potential-energy curves and on vibrationally specific, cumulative reaction probability densities.

Significance

These studies lead to a deeper understanding of fundamental collision processes in chemical kinetics and of quantum effects in kinetics and energy transfer.

Future Plans

Future work will include calculations of atom-diatom reactions and HF-HF energy transfer collisions.



Density of reactive states for fixed-energy, fixed-total-angular-momentum ensembles as a function of energy E , for total angular momentum $J = 0$. The solid curve was obtained by differentiating the calculated accurate quantum mechanical cumulative reaction probability (summed over all initial and final states open at the given E). The dashed curve is a fit based on ten quantized thresholds treated as one-dimensional effective parabolic barriers. The feature numbers correspond to quantized transition states. This figure shows that the suprathreshold reactivity is globally controlled by quantized transition states up to very high energies.

Hot Gas Ingestion by STOVL Aircraft

Pratap Vanka, Principal Investigator

Co-investigators: D. K. Tafti and W. Pegeus

University of Illinois, Urbana-Champaign

Research Objective

The goal of this research project is to develop an efficient computational capability to study hot gas ingestion by short takeoff and vertical landing (STOVL) aircraft during hover and landing in ground proximity. The computational procedure is based on using multigrid techniques to provide efficient convergence to a steady-state flow field.

Approach

The three-dimensional steady form of the Navier-Stokes equations is solved by a finite-volume technique using a multigrid-based solution procedure. The momentum and continuity equations are solved in a coupled manner. A $k-\epsilon$ model is used for modeling the turbulent fluxes.

Accomplishment Description

A previously developed multigrid-based code was extended to handle embedded obstacles and mass and momentum injections in order to simulate the aircraft fuselage and the lift jets. An equation for enthalpy was added to calculate the temperature field. Several validation calculations were completed, to assess the code and the turbulence model. Initially the impingement of a vertical single jet was calculated and the results compared with experiment. Subsequently, the impingement of twin jets and the formation of an upwash fountain were calculated and the results compared with data from McDonnell Research Laboratories. Finally, the code was used to simulate the fuselage of an aircraft and the lift jets. Four test calculations were completed, with variations in the height of the lift jets from the ground and in the headwind

speed. The results were compared with previous calculations from NASA Lewis. In all these calculations, a fine grid with up to 210,000 cells was used. The final calculation converged in approximately 140 iterations and required approximately 35 minutes on a Cray-2 and 14 megawords of memory. Compared with a previously used single-grid method, the CPU time was reduced by a factor of 10.

Significance

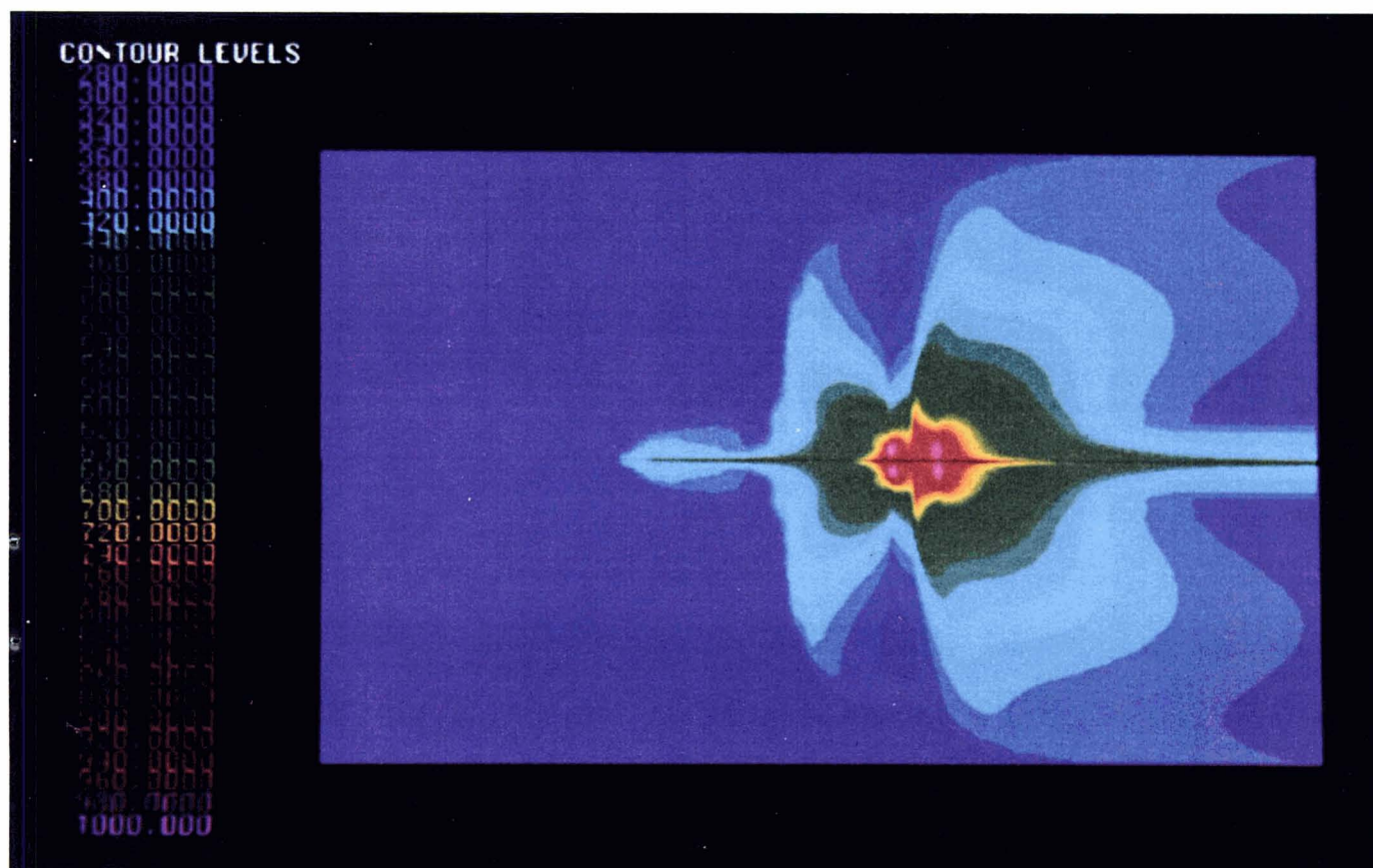
The proposed computational procedure is useful in understanding hot gas ingestion by STOVL aircraft and in conducting parametric studies to modify the design of the inlets. Parametric studies are in progress to investigate the effects of jet height, jet Mach number, and headwind speed.

Future Plans

The computer program is currently being extended to nonorthogonal grids. After this extension, realistic shapes of the aircraft can be simulated and more accurate parametric tests can be conducted.

Publications

1. Tafti, D. K., and Vanka, S. P. "Hot Gas Environment Around STOVL Aircraft in Ground Proximity, Part 2: Numerical Study." Presented at the 26th AIAA/ASME/ASEE Joint Propulsion Conference, Orlando, FL, July 1990.
2. Pegeus, W. J., and Vanka, S. P. "Numerical Study of Twin-Jet Impingement Upwash Flow." Presented at the ASME Forum on Turbulent Flows, Toronto, Canada, June 1990.



Ground-plane temperature distribution for $H/D_j = 4.0$ and $U/V_j = 0.03$.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Calculation of Hypersonic Flows with Strong Surface Blowing

Bram vanLeer, Principal Investigator

Co-investigators: T. C. Adamson, A. F. Messiter, N. D. Carter, and M. D. Matarrese

The University of Michigan

Research Objective

A possible method of controlling hypersonic flight vehicles is to generate the required control forces by modifying the surface pressure distribution through the use of boundary-layer blowing. The ultimate objective of our work is to produce a computer code that can accurately predict the resulting pressure distribution for a given injection pattern.

Approach

A higher order Godunov-type finite-volume approach is used to discretize the two-dimensional Navier-Stokes equations. Interface values of the state quantities are reconstructed using a monotone interpolation technique suggested by Koren, based on van Leer's kappa scheme. The interface inviscid fluxes are computed using Roe's upwind-biased flux-difference splitting techniques. Viscous fluxes are calculated using central differencing. Proper treatment of the downstream boundary condition is achieved using a modified form of Hedstrom's (1970) time-dependent characteristic conditions. The time-differencing algorithm is an explicit multistage scheme with optimized short-wave damping.

Accomplishment Description

Analytical similarity solutions exist for strong blowing along a flat plate and along a wedge with a continuous inverse-square-root injection-velocity distribution. Such solutions were used to validate the computer code during its development. The computer code produced results that compared very well with the analytical predictions for cases in which viscous

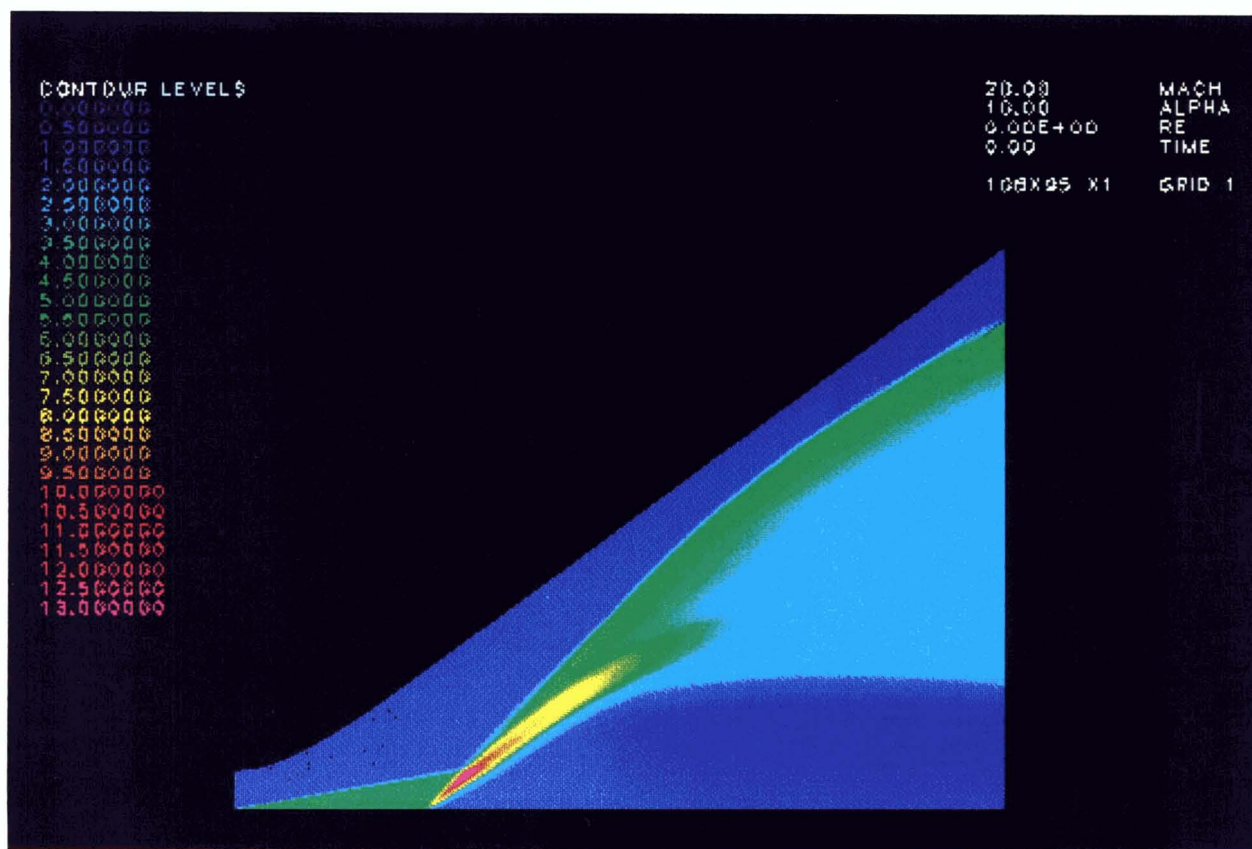
effects were negligible. The accompanying figure shows a density contour plot for a case in which free-stream-density air is uniformly injected along a wedge from $X = 0.25$ to $.5$. The code is currently being validated for flows in which viscous effects are important. It has also been modified to handle nonreacting multicomponent flow so that the effects of foreign gas injection can be studied. A typical viscous multicomponent flow calculation requires about 50 Cray Y-MP minutes and one megaword of memory.

Significance

The generic problem of boundary-layer blowing in hypersonic flows has many applications beyond vehicle control. Similar flow fields are found in the areas of transpiration cooling of hypersonic vehicles and the introduction of fuel into the combustion region of a scramjet. Successful simulation of such flow fields is vital to the understanding and efficient application of boundary-layer blowing in hypersonic flows.

Future Plans

Once the computer code has been verified to accurately calculate viscous flow fields for which analytical solutions are known, it will be used to model flows for various complex injection distributions. Also, it will be extended to handle real-gas effects and equilibrium flow.



Density contours for a wedge along which free-stream-density air is uniformly injected from $X = 0.25$ to 0.5 ; $M = 20$, $\alpha = 10^\circ$, $Re = 0.0$, time = 0.0 .

ORIGINAL PAGE
COLOR PHOTOGRAPH

High Reynolds Number, Transonic, Viscous Flow over Aircraft Components

Veer N. Vatsa, Principal Investigator

Co-investigator: Bruce W. Wedan

NASA Langley Research Center

Research Objective

To develop an efficient and robust numerical procedure for computing high-Reynolds number, viscous flows over aircraft components.

Approach

A multistage Runge-Kutta time-stepping scheme with a multigrid acceleration technique is used for computing the steady-state solutions of the thin-layer Navier-Stokes equations. Baldwin-Lomax and Johnson-King models are used for turbulence closure.

Accomplishment Description

A finite-volume numerical scheme was developed for computing high-Reynolds number viscous flows over aircraft components. The efficiency of the scheme was enhanced significantly through a multigrid acceleration technique. An order-of-magnitude reduction in computational time was achieved because of the multigrid technique. The code was used to solve flow over transport wings, fighter wings, wings-in-tunnel, and wing/fuselage configurations. An exhaustive grid-refinement study was conducted for flow over a transport-type wing, namely, the ONERA M6 wing. Because of the improved efficiency of the code, essentially grid-converged solutions for transonic flows were obtained for the first time using the thin-layer Navier-Stokes equations. The computational time required for obtaining a converged solution was found to vary linearly with the number of mesh points used, which indicated that the multigrid algorithm was working as expected. The error estimates for force coefficients (lift, drag, and pitching-moment) based on extrapolated infinite mesh solutions are shown in the accompanying figure. Note that the drag coefficient would require the most resolution for a specified level of accuracy.

Significance

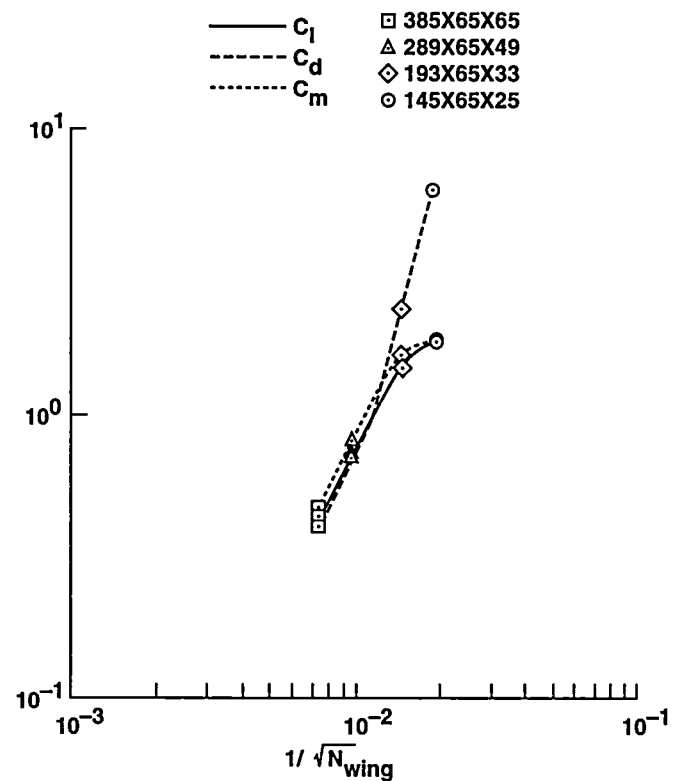
A numerical procedure of the type described here can be used to obtain accurate and reliable solutions to viscous flow problems of interest, since it will produce essentially grid-converged solutions in a timely manner.

Future Plans

The method described here will be generalized to accommodate block-structured grids so that flow over complex aircraft configurations can be computed.

Publications

Vatsa, V. N., and Wedan, B. W. "Development of an Efficient Multigrid Code for 3-D Navier-Stokes Equations." AIAA Paper 89-1791, 1989.



The effect of tangential grid spacing on force coefficients.

Tactical Missile Aero-Propulsion Interaction

Bill J. Walker, Principal Investigator

Co-investigators: C. D. Mikkelsen, K. D. Kennedy, K. L. Cornelius, and M. E. Vaughn, Jr.

U.S. Army Missile Command—Redstone Arsenal

Research Objective

To develop the capability to model (1) missile plume flow fields, including aerodynamic flow separation of the missile afterbody boundary layer, which is caused by the high pressures at the rocket nozzle exit plane; (2) the recirculation, mixing, and chemical reaction in the base region; (3) the mixing, shock structure, and afterburning of the fuel-rich rocket exhaust in the near-plume region; and (4) the mixing in the far-plume region.

Approach

The technical approach entailed the simultaneous solution of the fluid-dynamic conservation equations; the turbulence model equations; the two-phase, gas-particle interaction equations; and the chemical kinetics equations, using proven numerical methods of computational fluid dynamics.

Accomplishment Description

A full, three-dimensional, Navier-Stokes flow-field solver complete with two-equation turbulence models, finite-rate chemistry, and a two-phase flow capability was developed

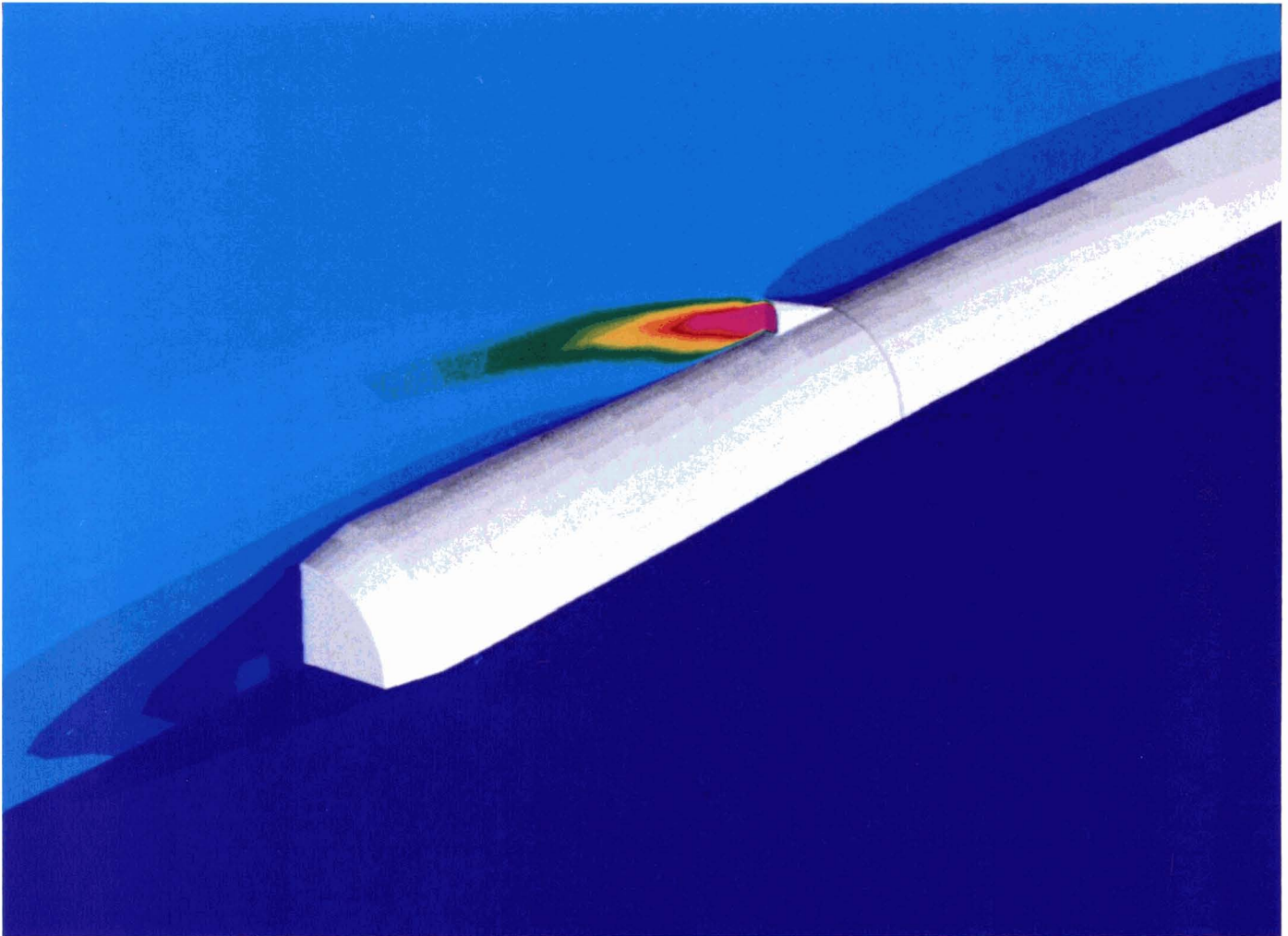
and demonstrated for a realistic, three-dimensional, tactical-missile aero-propulsion interaction case. The accompanying figure shows the flow-field velocity magnitude for a subsonic missile with bifurcated propulsive nozzles and a large blunt base. The case was run using blocking to handle high-resolution grids for the forebody, the aftbody with bifurcated nozzles, and the wake region. This case required 13.5 Cray hours and 40 megawords of CPU memory.

Significance

The basic method now exists to numerically model and predict the complex, chemically reacting and two-phase flow fields for missiles with propulsive rocket exhaust plumes. This method, when validated, will provide invaluable missile design information that cannot be acquired in wind tunnels because of capability, safety, or cost.

Future Plans

Efforts will focus on validation of the model with known experimental data.



The flow-field velocity magnitude for a subsonic missile.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Development of Unstructured and Nonequilibrium Chemistry Upwind Algorithms for Hypersonic Flows

Robert W. Walters, Principal Investigator

Co-investigators: Bernard Grossman, P. Cinnella, Andrew S. Godfrey, William D. McGrory, and David C. Slack
Virginia Polytechnic Institute and State University

Research Objective

The objective of this work is the prediction of hypersonic flow fields, including the effects of finite-rate chemistry and non-equilibrium thermodynamics. In addition, unstructured flow solvers for complex geometry modeling and automatic mesh adaptation are being developed.

Approach

A three-dimensional, implicit, upwind Navier-Stokes code with generalized chemistry and vibrational nonequilibrium thermodynamics models is used to predict the flow field about hypersonic vehicles in propulsion systems.

Accomplishment Description

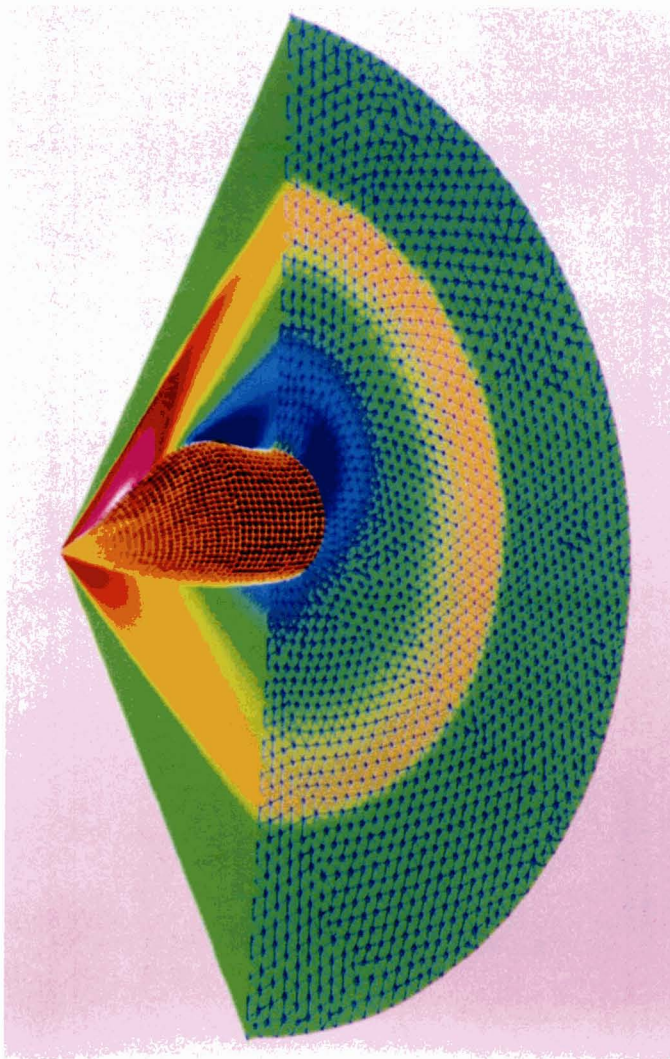
In the previous operational year, we developed a three-dimensional, explicit Euler code with finite-rate chemistry. This year we significantly extended the General Aerodynamic Simulation Program (GASP) and performed code validation experiments. The most significant enhancements included the extension to the full Navier-Stokes equations, the thin-layer Navier-Stokes equations, and the parabolized Navier-Stokes equations. Space-marching was added so that the user can partition a domain into "elliptic" and "parabolic" regions. Implicit time-integration schemes with fully coupled chemistry were added with several options (e.g., re-use of the Jacobian matrices and vectorization over planes), and a multiblock capability was included for running on composite grids. Several chemistry models were added and tested during this operational year. Many code validation experiments were performed over a wide range of Mach numbers and flight conditions in order to test the features of GASP. A typical calculation with GASP is 10 hours of Cray Y-MP time with 20 megawords of memory. In October, a beta version was released to the National Aero-Space Plane contractors for testing and feedback. In addition, work on high-order-accurate ENO schemes and unstructured flow solvers continued. Automatic mesh adaptation and unstructured space-marching algorithms were developed during this period. The figure shows an example of a space-marching calculation on a hybrid structured/unstructured grid. Pressure contours in the symmetry planes and the exit plane of an experimentally tested analytic forebody are shown in the figure along with the grid topology. These emerging technologies will be included in a later release of GASP.

Significance

The design of hypersonic vehicles relies heavily on computational fluid dynamic simulations. The need to accurately model the complex chemistry and physics in these high-enthalpy external and internal flow fields requires the development and refinement of three-dimensional Navier-Stokes codes. Run times for these complex codes for realistic configurations are large; thus mesh adaptation is likely to play an increasing role, since it offers the hope of efficient placement of elements.

Future Plans

The GASP code will be extended to include a generalized indexing scheme such that calculations on both structured and unstructured grids can be performed. Characteristic-based algorithm development, including multiple translational temperatures for more accurate modeling of free-electron transport, will be completed and implemented. Nose-to-tail calculations, including the simultaneous simulation of both internal and external flows, will be performed.



Grid topology and pressure contours of a space-marching calculation.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Navier-Stokes Predictions of Hypersonic Chemically Reacting Flows

Jong H. Wang, Principal Investigator
Co-Investigators: David Yeh and Mike George
Rockwell International, North American Aircraft Division

Research Objective

To validate Navier-Stokes methods for predicting hypersonic chemically reacting flows.

Approach

Rockwell's unified solution algorithms (USA) code is applied to existing combustor-flow test cases to validate the code and to provide direction for future code development.

Accomplishment Description

A two-dimensional hypersonic-combustor-flow test with tangential hydrogen injection was investigated numerically. A mixing-length turbulence model and 1- and 18-step chemistry models were used to simulate the flow. A grid system containing 23,215 points was used. Solution times of 3.1 and 8.4 seconds on the Cray Y-MP computer were required per iteration for the 1- and 18-step models, respectively. The required memory was slightly over 2 megawords for the 18-step model and the solution took about 3000 to 5000 iterations for convergence. Solution strategies for speeding up the convergence rate were established. Also, it was concluded that the

numerical results were very sensitive to the combustor-inlet flow profiles. The predicted wall static pressure, heat transfer rate, and combustor-exit pitot pressure profiles were in good accord with experimental data, as was the location of the ignition point (which is indicated by the sudden rise of the combustor pressure). The predicted contours for pressure, temperature, H₂O mass fraction, H₂ mass fraction, and particle traces (adjacent to the hydrogen injection inlet) are shown in the accompanying figure.

Significance

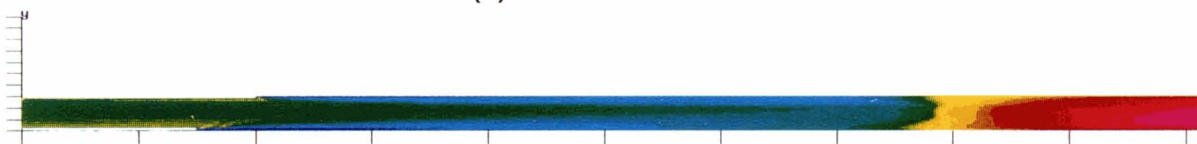
The design of an efficient hypersonic combustor is essential to the success of the National Aero-Space Plane and air-breathing hypersonic vehicles. The complexity of the flow physics and the need to obtain efficient mixing and combustion require an accurate and robust computational tool.

Future Plans

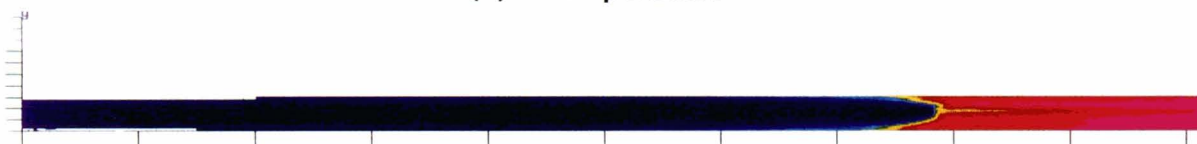
This research will be extended to study realistic three-dimensional combustor flows with various hydrogen injection angles.



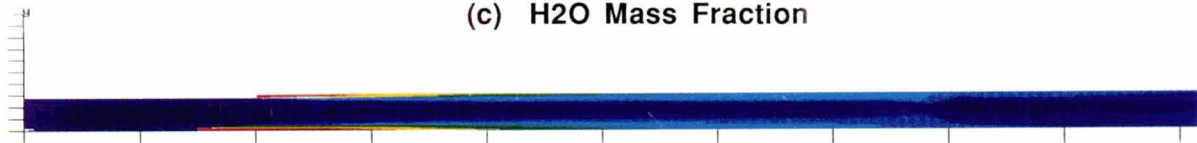
(a) Pressure



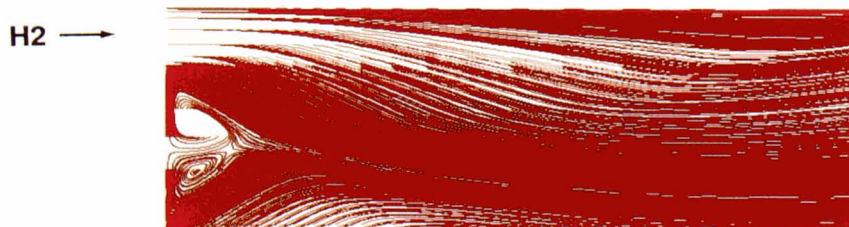
(b) Temperature



(c) H₂O Mass Fraction



(d) H₂ Mass Fraction



(e) Particle Traces Behind Backward Facing Step

Simulation of Three-Dimensional Flow in a Transonic Fan Rotor

Kurt F. Weber, Principal Investigator

Co-investigator: Dale W. Thoe

Allison Gas Turbine Division, General Motors Corporation

Research Objective

The objective of this work is to develop and validate a three-dimensional Navier-Stokes code that includes tip clearance modeling, for prediction of turbomachinery flows. The analysis will provide a detailed description of the flow through compressor rotors. Special emphasis is placed on the simulation of the flow field for the NASA transonic fan rotor designated Rotor 67.

Approach

A modified version of the NASA Ames finite-difference code ARC3D is applied to the solution of compressor and fan-rotor flow fields using body-conforming O-type grids in the blade passage and an embedded H-type grid in the tip-clearance region. Code-validation and grid-independence studies are performed for Rotor 67.

Accomplishment Description

An operational, three-dimensional, Navier-Stokes turbomachinery code was developed for use with O- and C-type grids. Modifications were made to ARC3D to include internal turbomachinery boundary conditions. Terms were added to the code to account for blade-row rotation, and to account for local metric invariant errors that arise when the strong conservation form of the equations is used. Fully viscous solutions on a fine O-type grid (376,309 points) were obtained for Rotor 67 at the peak-efficiency and near-stall operating points. These solutions show good agreement with laser Doppler anemometer measurements. The accompanying figure shows the static pressure contours for the O-grid solution at the near-stall

operating point. This solution used 18 megawords of storage and required 4.4 Cray-2 hours for 1000 iterations.

Significance

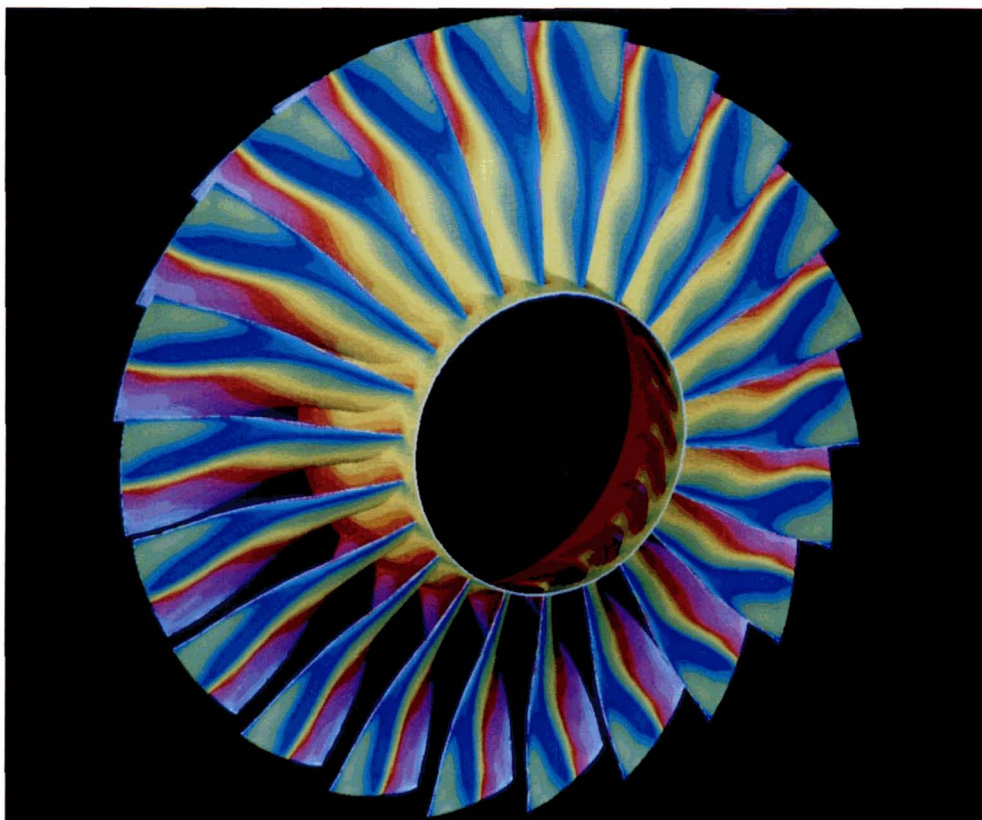
Historically, aerodynamic design improvements to aircraft gas-turbine-engine compressors and fans have depended on empirical correlations derived from extensive testing. Although this has produced lighter, more efficient compressors, it is unlikely that major improvements in performance can be achieved without great expense using this approach. Accurate modeling of the flow field within compressor blade rows will provide the information currently needed to improve compressor performance. It is also an important step toward developing a model to simulate the complete compressor flow field.

Future Plans

The O-grid code will be used as a starting point for a Navier-Stokes code that includes tip-clearance modeling. A fine H-type grid will be embedded in the tip-clearance region. The blade-passage and tip-clearance flow will be solved simultaneously using the Chimera composite grid scheme. This scheme involves the PEGASUS and F3D codes developed at NASA Ames and Arnold Engineering Development Center to solve the flow fields about complex geometries with multiple grids.

Publications

Weber, Kurt F.; Thoe, D. W.; and Delaney, R. A. "Analysis of Three-Dimensional Turbomachinery Flows on C-type Grids Using an Implicit Euler Solver." ASME Paper 89-GT-85, 1989.



Static pressure contours on the NASA Rotor 67 at the near-stall operating point.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Winged Entry Vehicle Computations

Kenneth J. Weilmuenster, Principal Investigator
Co-investigator: Francis A. Greene
NASA Langley Research Center

Research Objective

The objective of this work is to develop an integrated and systematic approach to the analysis of the aerothermal environment of a winged entry vehicle using state-of-the-art grid generation and computational fluid dynamics software. This analysis will extend over the hypersonic flight regime and require analysis that models the reacting-gas chemistry associated with hypersonic flight.

Approach

The three-dimensional Euler equations written in generalized coordinates and including perfect-gas as well as reacting-gas chemistry models are solved on a volume grid generated using software developed at the Langley Research Center.

Accomplishment Description

An existing three-dimensional, total-variation-diminishing (TVD), Navier-Stokes code was modified to create an Euler solver. Also, the TVD aspects of the code were modified to take into account the thermodynamics of a general chemically reacting gas. Currently, the code includes the Tannehill and Vinokur descriptions of equilibrium air chemistry and the thermodynamics of CF₄—a wind tunnel test gas and a perfect-gas option. The code was used to determine the aerodynamics of a proposed personnel launch system (PLS) configuration, for comparison with wind tunnel data at Mach numbers of 6 and 10. In addition, the code was used to determine the PLS

aerodynamics at several points along the hypersonic portion of the nominal PLS entry trajectory. Each solution of the PLS flow field requires 12 Cray-2 hours and 40 megawords of memory. During the NAS 1989-90 operational year, 448 Cray-2 hours were used for this project.

Significance

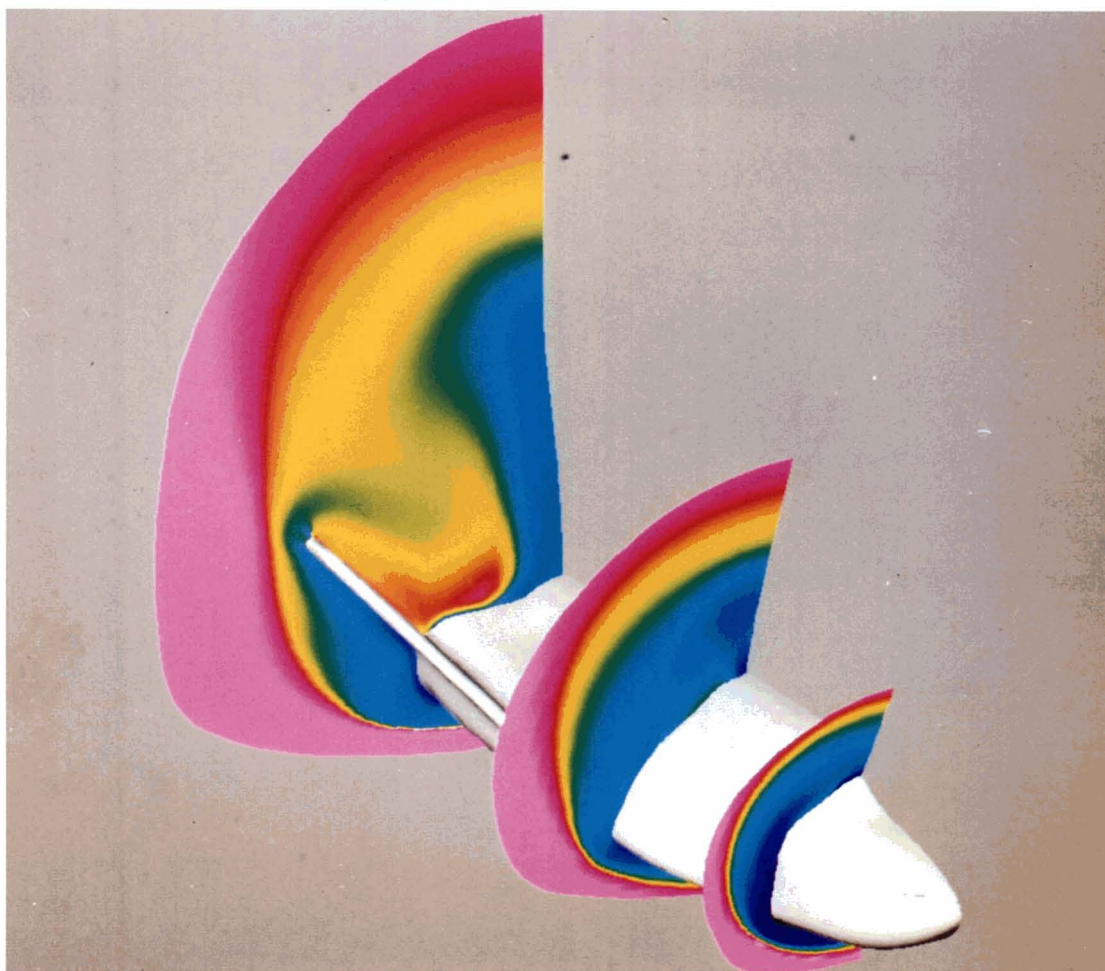
Computed and measured aerodynamics data at Mach numbers of 6 and 10 for a perfect-gas flow were in good agreement, as were the measured and computed aerodynamics data for Mach 6 flow using a CF₄ test gas. Based on the good agreement with measured data from ground-based facilities, computations were made at points along a nominal PLS trajectory. As a result of these computations, it has been determined that the PLS aerodynamics are only slightly affected by reacting-gas chemistry and that the vehicle trims at its maximum L/D in the hypersonic flight regime.

Future Plans

The NAS facility will continue to be used to support Euler computations for the PLS configuration. During the 1990-91 operational year, viscous solutions for flow over the PLS will be generated at selected wind tunnel and flight conditions.

Publications

Weilmuenster, K. J.; Smith, R. E.; and Greene, F. A. "Assured Crew Return Vehicle Flowfield and Aerodynamic Characteristics." AIAA Paper 90-0229, Jan. 1990.



Mach number contour plots for a personnel launch system vehicle; $M_\infty = 10$, $\alpha = 25^\circ$, $\gamma = 1.4$.

Optical Properties of Nonspherical Particles in Planetary Atmospheres

Robert A. West, Principal Investigator
Jet Propulsion Laboratory/California Institute of Technology

Research Objective

The ultimate objective of this work is to interpret spacecraft and ground-based observations of the radiometric and polarimetric properties of the atmospheres of the planets and comets. Although previous analyses relied on Mie scattering for spherical particles, Mie scattering calculations are inappropriate for many applications, especially for studies of the polarization of the nonspherical ice crystals and photochemical haze particles that form the clouds and aerosol layers of the outer planet atmospheres.

Approach

The optical properties (extinction, absorption, and scattering cross sections, and all elements of the phase matrix) are calculated with a discrete dipole approximation code, in which an arbitrarily shaped particle is modeled as an array of point dipoles on a lattice grid. A matrix equation describing the interaction of each dipole with the incident electromagnetic wave and with the fields of all the other dipoles is solved with a steepest-descent conjugate gradient method, to produce the radiated field of the particle.

Accomplishment Description

During the past year I completed a study of the optical properties of colloidal particles grown by diffusion-limited aggregation (DLA). This study was motivated by observations of the forward-scattering intensity of aerosols in Titan's atmosphere, made by the Voyager 1 imaging instrument, and by observations of the polarization of light scattered from the atmosphere, made by the Pioneer and Voyager photopolarimeter instruments. Early attempts to fit both the high polarization near a scattering angle of 90° and the strong forward scattering of Titan's haze particles with distributions of spheres failed. The strong polarization required the particle radius to be

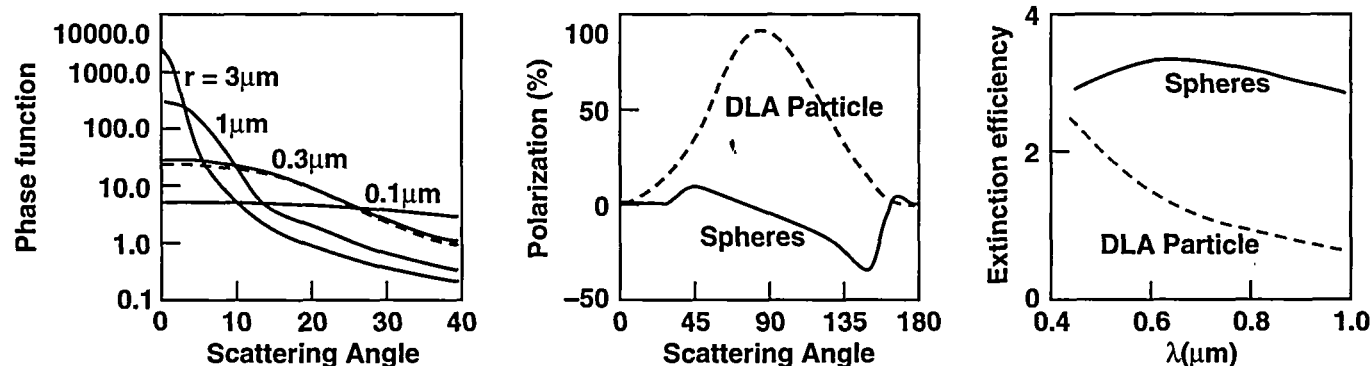
approximately $0.1\text{ }\mu\text{m}$ or smaller, while the phase function asymmetry required the particle radius to be about $0.3\text{ }\mu\text{m}$.

Significance

Aggregate particles were found to be abundant in a recent laboratory simulation of Titan's haze particles by Bar-Nun et al. (*J. Geophys. Res.* 93 (1988): 8383-8387). The calculations showed that these particles do indeed have both strong polarization and strong forward scattering as observed for Titan's particles, and thus a long-standing puzzle was solved. The implications of this finding are relevant to particle microphysical models and chemical evolutionary models of Titan's haze and surface composition. The aggregate particles have a much smaller density than equivalent-radius spheres, and their sedimentation velocity is smaller because of their larger geometric cross section for a given mass. Particle mass fluxes estimated from equivalent-radius spheres are about a factor of five or more larger than for aggregate particles. Microphysical models use this mass flux as a primary constraint on particle formation rates and surface deposition rates, and will have to be revised by the same factor. These results also have implications for experiments planned for the Huygens probe. Experiments that rely on Mie theory to relate scattered intensity to particle size, as some optical particle spectrometers do, will give erroneous values for particle size since the scattering efficiency of colloidal particles differs by a large factor from the scattering efficiency of equivalent-radius spheres. A paper is being prepared for submission to *Nature*.

Future Plans

Investigation of the optical properties of DLA particles will continue, with emphasis on understanding how the microstructure of the particle affects its optical properties. Studies of simulated comet grains are also under way, directed by collaborator Martha Hanner of JPL.



The optical properties of spheres are compared with those of particles formed by DLA. The left-hand figure shows the forward-scattering part of the phase function. The solid curves are for spheres of different radii, and the dashed curve is for a DLA particle with a projected cross-sectional area approximately equal to that of a sphere with radius $0.3\text{ }\mu\text{m}$. The shape of the forward-scattering peak provides a sensitive diagnostic of the particle projected area and is insensitive to particle shape. Other optical properties, such as the linear polarization and extinction efficiency, are sensitive to particle shape, as the center and right-hand figures demonstrate. Measurements of the optical properties of aerosols in Titan's atmosphere show modest forward scattering, and strong polarization near a scattering angle of 90° , which is indicative of DLA particles, not of spheres.

Application of Unstructured Grids to Shuttle Computational Fluid Dynamics

Thomas C. Wey, Principal Investigator
Co-investigator: Chien-Peng Li
Lockheed Engineering and Sciences Company/NASA Johnson Space Center

Research Objective

To develop a computational fluid dynamics capability to predict the high-speed flow around a realistic Space Shuttle configuration, while emphasizing the accuracy, efficiency, and affordability of Space Shuttle designs.

Approach

A three-dimensional structured/unstructured grid generation program and its corresponding Euler equation solver are coded for this project.

Accomplishment Description

A new and simple advancing-front technique for unstructured-grid generation was developed. A new generalized upwind-biased interpolation scheme was derived to achieve second- or third-order accuracy of the Euler equation for the unstructured meshes. The computation for the flow at Mach number 7.32 and a 35.1° angle of attack needed about 9 Cray-2 hours and 20 megawords of memory. The computation needed about

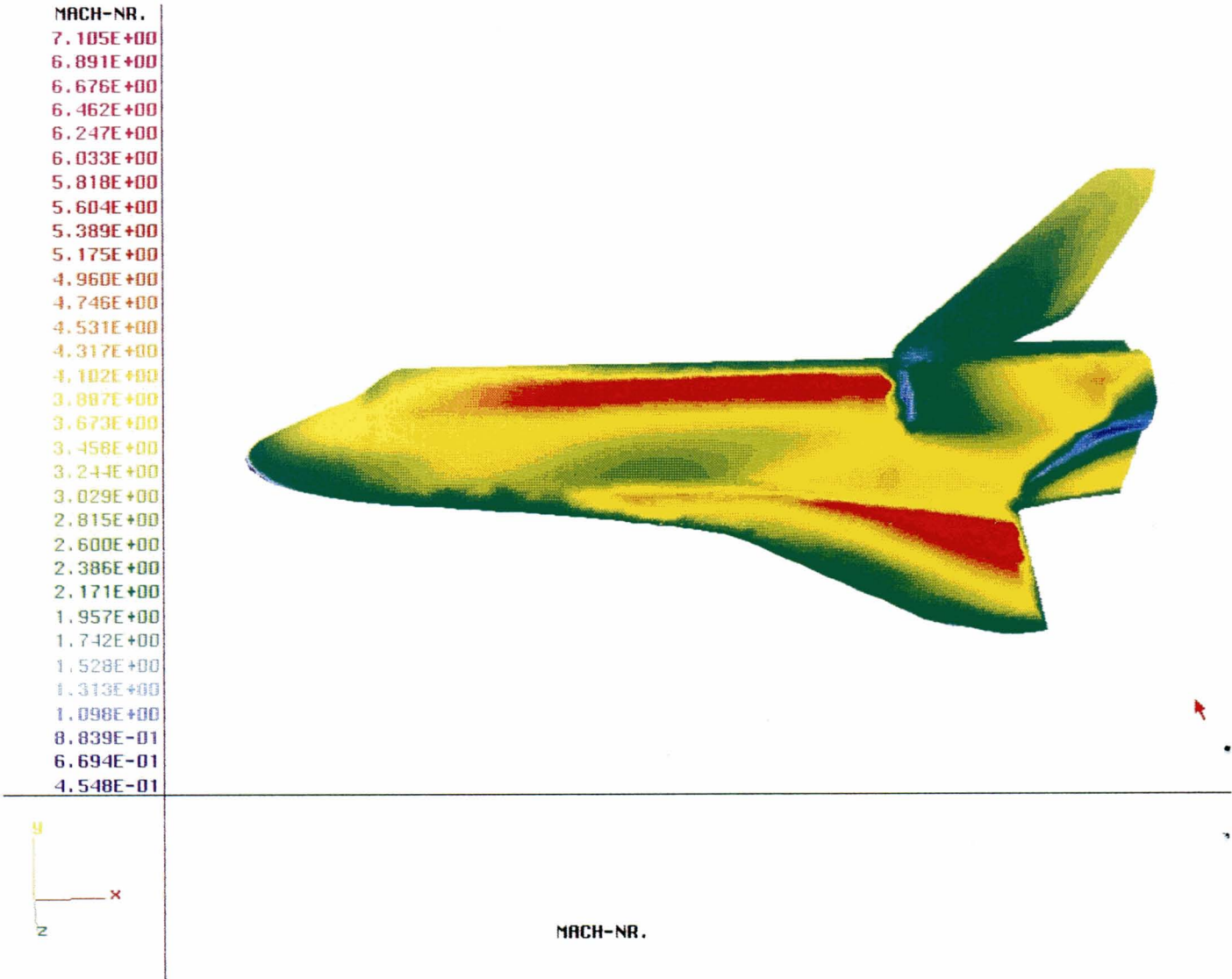
2400 Cray-2 seconds and 5 megawords of memory to generate a set of grids that consists of 109,866 nodes and 641,276 tetrahedra.

Significance

The memory required for this grid generation is significantly smaller than that for other unstructured-grid generation methods. Because of the nature of the advancing-front scheme, a major part of the grid generation program runs in the scalar mode. Therefore the grid can be generated using a less powerful scalar computer, such as a VAX computer.

Future Plans

It is not practical at this time to discretize the entire flow domain using unstructured grids. Hence the combination of structured and unstructured grids has been implemented in the flow solver. The effort to code a program to combine multiblock structured and unstructured grids and output the connectivity information between them is pending.



Surface Mach number contours of the orbiter, calculated with unstructured grids.

Shock Interactions in Rarefied Hypersonic Flows

Richard G. Wilmoth, Principal Investigator

Co-investigator: Bradford Sturtevant

NASA Langley Research Center/California Institute of Technology

Research Objective

To provide a basic understanding of rarefied flow effects on shock interactions in hypersonic flows, and to enhance the computational capability to treat the complex shock/vehicle interactions that take place at reentry speeds.

Approach

The direct-simulation Monte Carlo (DSMC) method is used to model the flow about various geometries in order to study a variety of shock/boundary and shock/leading-edge interactions. The simulation includes collision models that represent real-gas effects and nonequilibrium chemistry. An alternate computational approach involving finite-difference solutions of the Boltzmann equation using discrete-velocity models is also being pursued.

Accomplishment Description

Computational efforts for the DSMC method were focused on the use of multi-tasking on the Cray Y-MP and on calculations in support of research being conducted separately on parallel processing using hypercube architectures. Computations were up to three times faster when multi-tasking was used. Fine-grid solutions for a hypersonic channel flow were completed and the results used to investigate requirements for adaptive parallel domain decomposition on the Intel hypercube. The attached figure shows gray-shaded density contours for one such calculation, with the domain boundaries superimposed. Applications studies focused on bow-shock definition in transitional flows about spherical bodies representative of typical tethered satellites. For the Boltzmann equation studies, computations were completed for several one- and two-dimensional problems designed to determine the number of discrete velocities needed to achieve an

adequate accuracy for each problem. Velocity models using from 7 to 13 discrete values per component were used to compute normal shock solutions to accuracies of less than 1%. Typical computing resources needed for DSMC applications studies range from less than 1 hour with 1 to 2 million words up to 20 to 30 hours for problems requiring 4 to 8 million words.

Significance

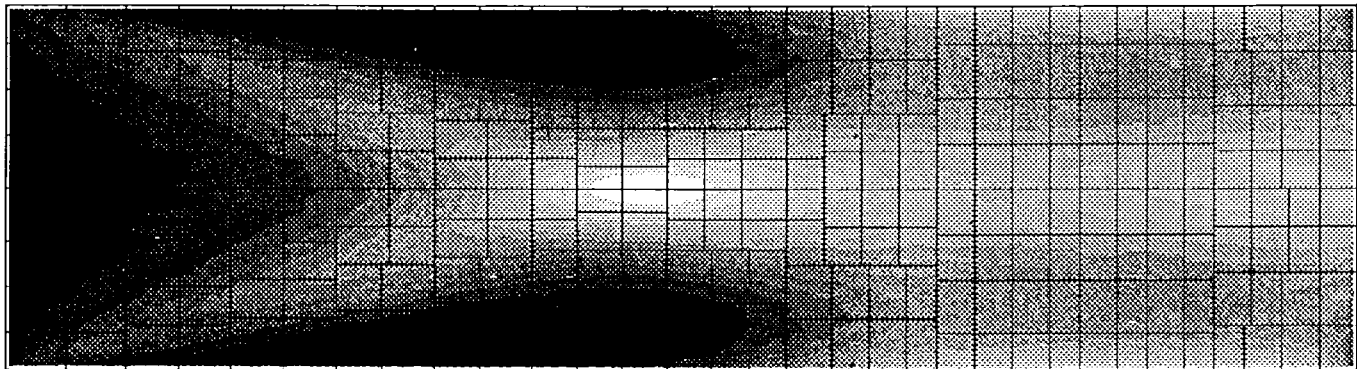
Parallel processing shows tremendous potential for reducing the computational times required by the DSMC method. Such reductions will be necessary for future three-dimensional simulations of full-scale vehicles in the transitional flow regime. The Boltzmann-type solutions also provide much-needed fundamental information for further theoretical validation of the DSMC method.

Future Plans

Work will be expanded on the use of vectorization and multi-tasking for the DSMC codes. A new method for three-dimensional simulations will be applied to more realistic geometries of full-scale vehicles. Work will continue on developing efficient parallel processing techniques on a variety of machine architectures.

Publications

1. Wilmoth, Richard G. "Interference Effects on the Hypersonic Rarefied Flow About a Flat Plate." *Progress in Astronautics and Aeronautics* 118 (1989).
2. Wilmoth, Richard G. "Direct Simulation Monte Carlo Analysis on Parallel Processors." AIAA Paper 89-1666, June 1989.



A DSMC simulation of a shock interaction in Mach 8 channel flow; the domain boundaries for parallel decomposition are superimposed on density contours.

Computational Fluid Dynamics Design of a Laminar-Flow Control Glove

Chung-Jin Woan, Principal Investigator
Co-investigator: Michael W. George
Rockwell International, North American Aircraft Division

Research Objective

The current objective of continuing research involves a theoretical investigation of the conditions under which a single linear program is guaranteed to solve for the global minimum of a large-scale, constrained, nonlinear global-optimization problem. This objective arose from the discovery that the global optimum is very often detected after the solution of a single linear program, but cannot often be verified as such until many further iterations are completed.

Approach

A three-dimensional Navier-Stokes code is used in conjunction with a compressible stability-analysis code to create a numerical approach. The approach is being applied to a modified F16xL, which will be flight-tested in April 1990. This NAS effort is an adjunct effort to the larger NASA/Rockwell cooperative program.

Accomplishment Description

A nonaligned patched-grid technique was developed and incorporated into an existing three-dimensional Navier-Stokes code to treat the wing-body-inlet combination of the modified F16xL fighter. A blocked grid was generated that had 21 blocks and a total of 759,181 grid points. A solution without suction was obtained. In the flow calculation, a continuing-block-solution approach was used. In this approach, the entire solution domain was divided into four main blocks in the streamwise direction, and the solution was calculated by marching from the most upstream block to the most down-

stream block. A time-marching scheme was used to compute the solution of each of the four main blocks. At the block interfaces, a supersonic outflow boundary condition was used for the upstream block and a supersonic inflow boundary condition was used for the adjacent downstream block. The accompanying figure shows the calculated pressure-coefficient contours on the modified F16xL surface. Also shown in the figure are the nonaligned grids at the block interface located at the middle of the canopy. The computation used 39 Cray-2 hours, and the largest block calculation required 18 megawords of memory.

Significance

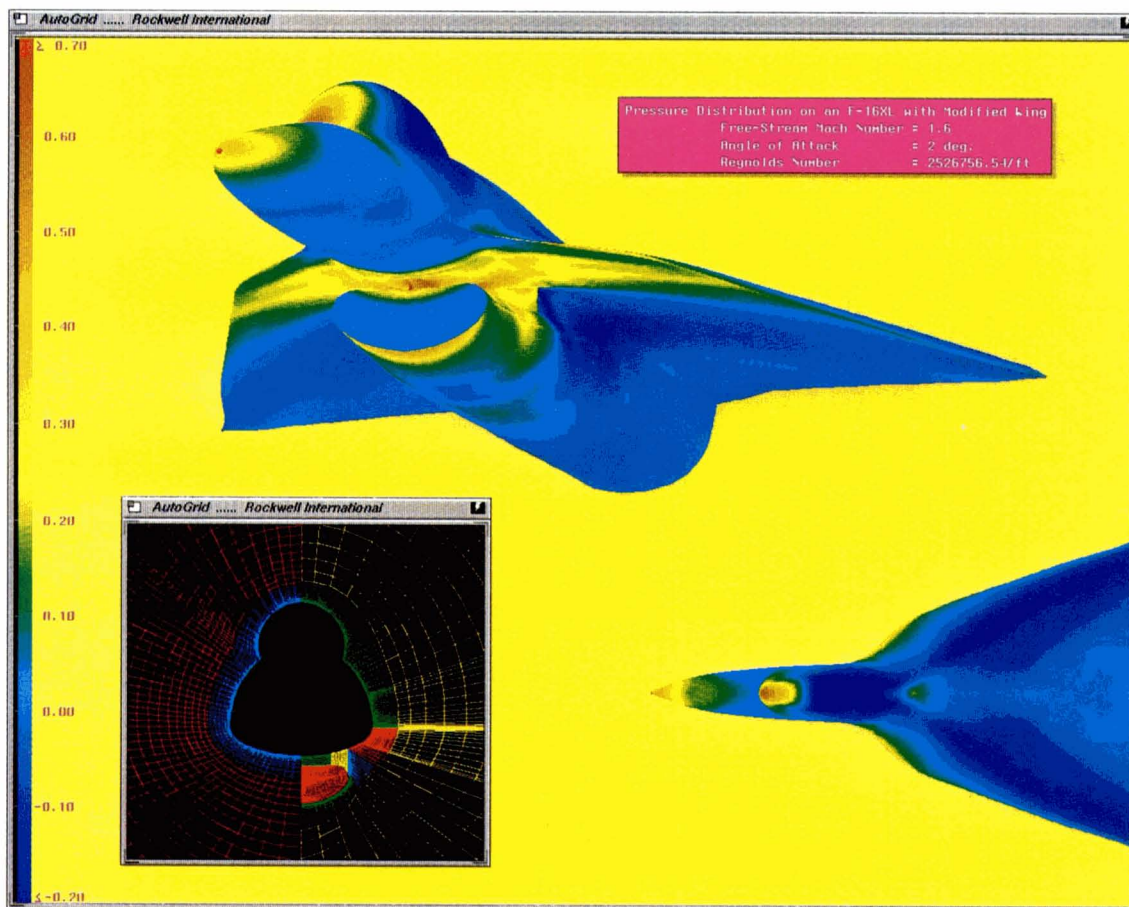
The generation of a numerical ability to predict transition, and the extension of laminar flow control to supersonic and hypersonic regimes, would benefit several national programs, including the National Aero-Space Plane.

Future Plans

Continued effort in calculating the flow about the modified F16xL fighter will focus on the cases with suction and on validating the supersonic laminar flow control concept by comparing calculated results with flight test data. The modified F16xL fighter will be flight tested in April 1990.

Publications

"CFD Validation of a Supersonic Laminar Flow Control Concept." An abstract to be submitted to the AIAA 29th Aerospace Sciences Meeting and Exhibit, Jan. 1991.



Pressure distribution on an F16xL fighter with a modified wing; $M_\infty = 1.6$, $\alpha = 2^\circ$, $Re = 2,526,756.54/ft$.

Numerical Simulation of Submarine Propulsion

Cheng-I Yang, Principal Investigator
David Taylor Research Center

Research Objective

To extend the capabilities of an existing Navier-Stokes code to compute the hull/propulsor interaction and thrust deduction of a submarine.

Approach

Viscous flow over a submarine hull with and without a propeller in operation are simulated. Computations are performed with the FMC1 code developed at NASA Langley Research Center. The method is based on an implicit high-resolution scheme for the three-dimensional, incompressible Navier-Stokes equations. The inviscid flux is discretized with TVD-like flux-difference splitting, and the viscous flux is discretized with central differencing. The computational grid is generated by a transfinite interpolation technique.

Accomplishment Description

The effect of the propeller is incorporated into the Navier-Stokes equation with an imbedded body-force distribution method; it includes both axial and tangential components. The differences between axial velocity components ($\Delta U_x/V_\infty$) with and without a propeller in operation at two streamwise locations are shown in the attached figure (x is the distance from the bow, L is the hull length, and r is the normal distance from the body surface). The propeller is located at $x/L = 0.983$.

The differences in pressure distribution on the after-portion of the hull are also shown in the figure. The agreement between the computed and the measured values is very good. From the results of the computation, the influence of propeller action can be detected up to about two propeller diameters upstream. Typical calculations on a grid of about 150,000 nodal points require about 1200 CPU seconds on a CRAY Y-MP, with 2 megawords of memory.

Significance

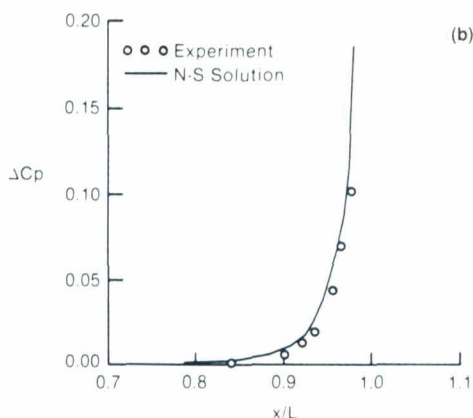
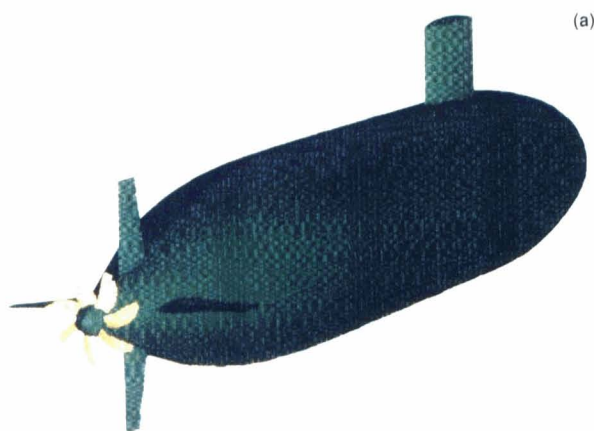
The design of an efficient propulsor system for an underwater vehicle requires a better understanding of hull/propeller interaction. It is demonstrated that essential information can be obtained by a computational fluid dynamics simulation.

Future Plans

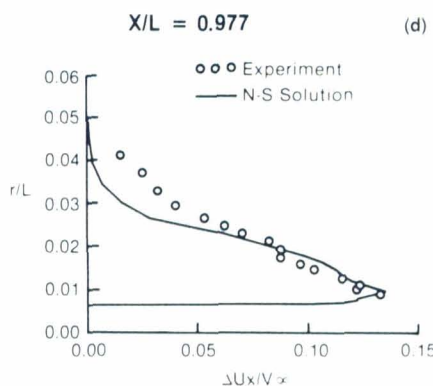
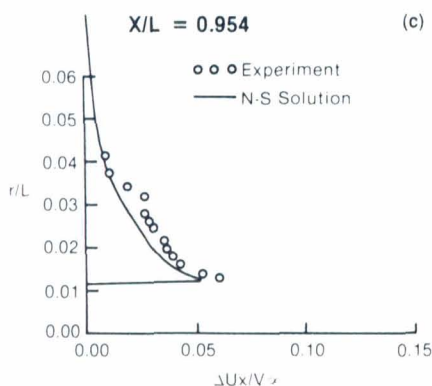
The present method will be used to study the viscous flow around a compound propulsor. The method will also be used to study the detailed flow pattern around a propeller blade tip.

Publications

Yang, C.-I.; Hartwich, P.-M.; and Sundrarum, P. "Numerical Simulation of Three-Dimensional Viscous Flow Around a Submersible Body." Presented at the Fifth International Conference on Numerical Ship Hydrodynamics, Hiroshima, Japan, Sept. 1989.



Measured and Computed Surface Pressure Decrease on Afterbody 1 Due to Propeller Suction



Measured and Computed Axial Velocity Increase on Afterbody 1 Due to Propeller Suction

(a) Submarine hull with propeller. Measured and computed (b) surface pressure decrease, (c) axial velocity increase at $x/L = 0.954$, and (d) axial velocity increase at $x/L = 0.977$ on afterbody 1, resulting from propeller suction.

Multistage Rotor/Stator Interaction in the Space Shuttle Main Engine Turbopump Turbine

Ruey-Jen Yang, Principal Investigator
Co-investigator: Shyi-Jang Lin
Rockwell International, Rocketdyne Division

Research Objective

To develop and validate a two-dimensional Navier-Stokes code for predicting multistage rotor/stator unsteady flow interactions, with emphasis on the Space Shuttle main engine.

Approach

The NASA Ames ROTOR-2 code, which was developed by M. M. Rai, is modified. The ROTOR-2 code is a two-dimensional Navier-Stokes code for single-stage turbomachinery. It uses multiple-zone technology and an iterative, implicit, third-order-accurate upwind algorithm.

Accomplishment Description

The upgraded ROTOR-2 code can handle rotor/stator interaction problems for arbitrary numbers of stages and airfoils. To validate the new code, a 1.5-stage, large-scale rotating rig (airfoil ratio 3:4:4) of the United Technology Research Center was simulated, and the results compared well with experimental data. For the numerical accuracy check, investigations were also carried out on the effect of thin-layer versus full

viscous calculation, the effect of temporal accuracy, and the effect of inner subiteration of the numerical scheme. A calculation for the 6-stage, low-pressure oxidizer turbopump axial turbine of the Space Shuttle main engine was also completed.

Significance

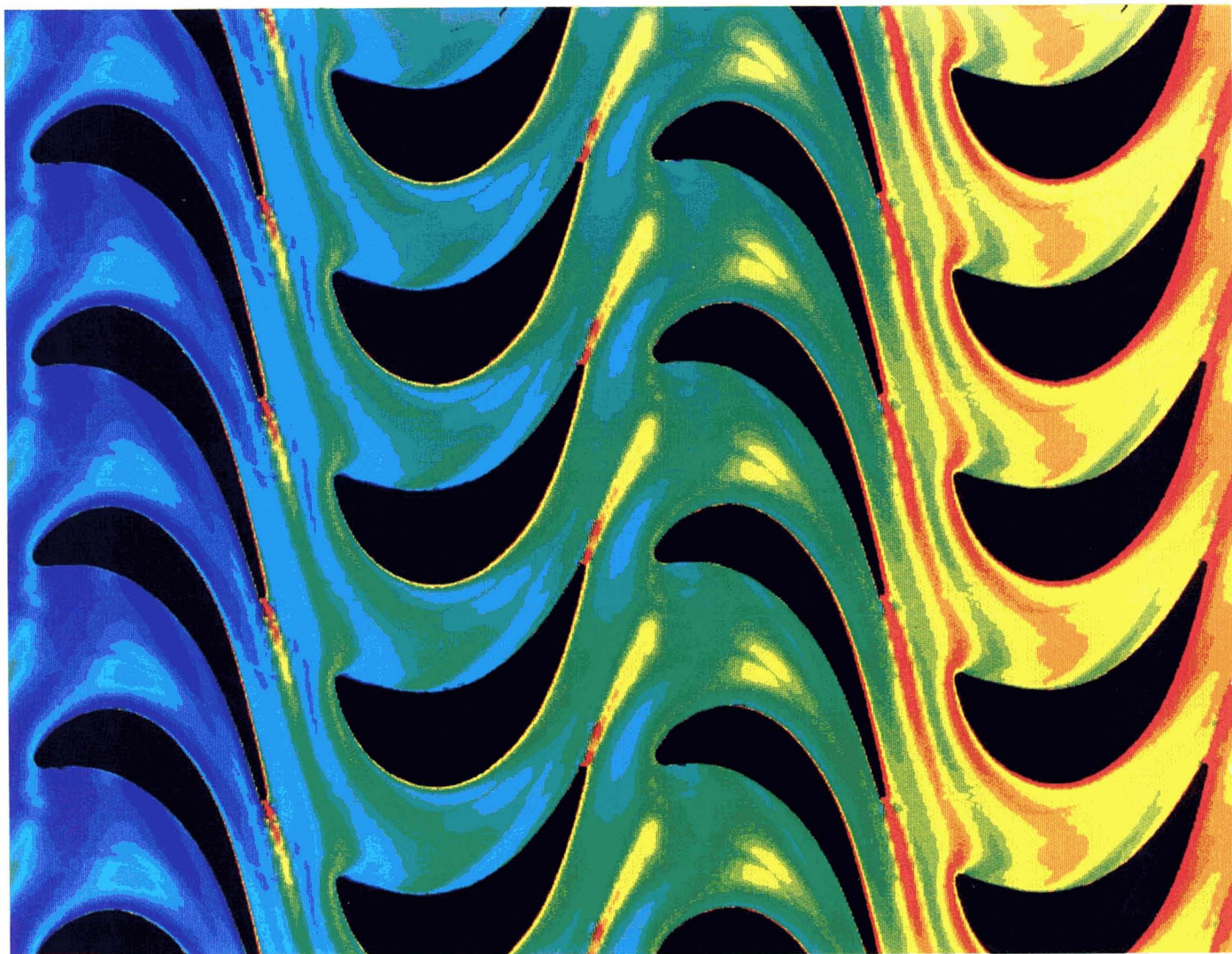
The use of multistage computations is necessary to obtain accurate transient flow information for multistage machines and, in particular, for those components along downstream stages. The accurate transient flow information is essential for providing dynamic loadings for structural model analysis.

Future Plans

The code will be further used to study supersonic, multistage, rotor/stator unsteady-flow interactions.

Publications

Yang, R.-J., and Lin, S.-J. "Numerical Solutions of 2-D Multi-Stage Rotor/Stator Unsteady Flow Interactions." *Proceedings of CFD Symposium on Aeropropulsion*. NASA Lewis Research Center, Apr. 1990.



Calculation of a multistage rotor/stator interaction.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Tilt-Rotor Download Prediction

Larry Young, Principal Investigator
Co-investigator: C. S. Lee
NASA Ames Research Center

Research Objective

To develop an accurate and efficient code to predict the tilt-rotor download in hover.

Approach

A low-order panel method is used. The rotor blades are modeled by an actuator disk with a known normal velocity distribution. The rotor wakes are simulated by a linear doublet in the streamwise direction. The wing is modeled by constant source and doublet panels. The separated wing wake is represented by constant doublet panels. The separation lines on the wing surface are assumed to be along the leading edge, wing tip, and trailing edge. The wake strength and the base pressure on the wing lower surface are determined by the Kutta condition for separated flows. Near-field correction in induced velocity for the close vortex/surface interaction is implemented to prevent the rotor wake from penetrating into the wing upper surface.

Accomplishment Description

The development of the download prediction was divided into stages corresponding to each component of the flow feature: (1) three-dimensional massive separated flow, (2) rotor simulation, and (3) vortex filament/wing surface interaction. Progress has been made on each of the topics. For massive separated flows, a calculation was performed and reported for a wing in uniform flow with a 90° angle of attack. For unsteady rotor wake simulation, curved vortex elements were used for the rotor wake. For close vortex/surface interaction, computa-

tional schemes such as near-field correction, wake element redistribution, and wake deflection near surfaces were developed. Each flow computation required approximately 1 hour of Cray-2 time and 2 megawords of memory.

Significance

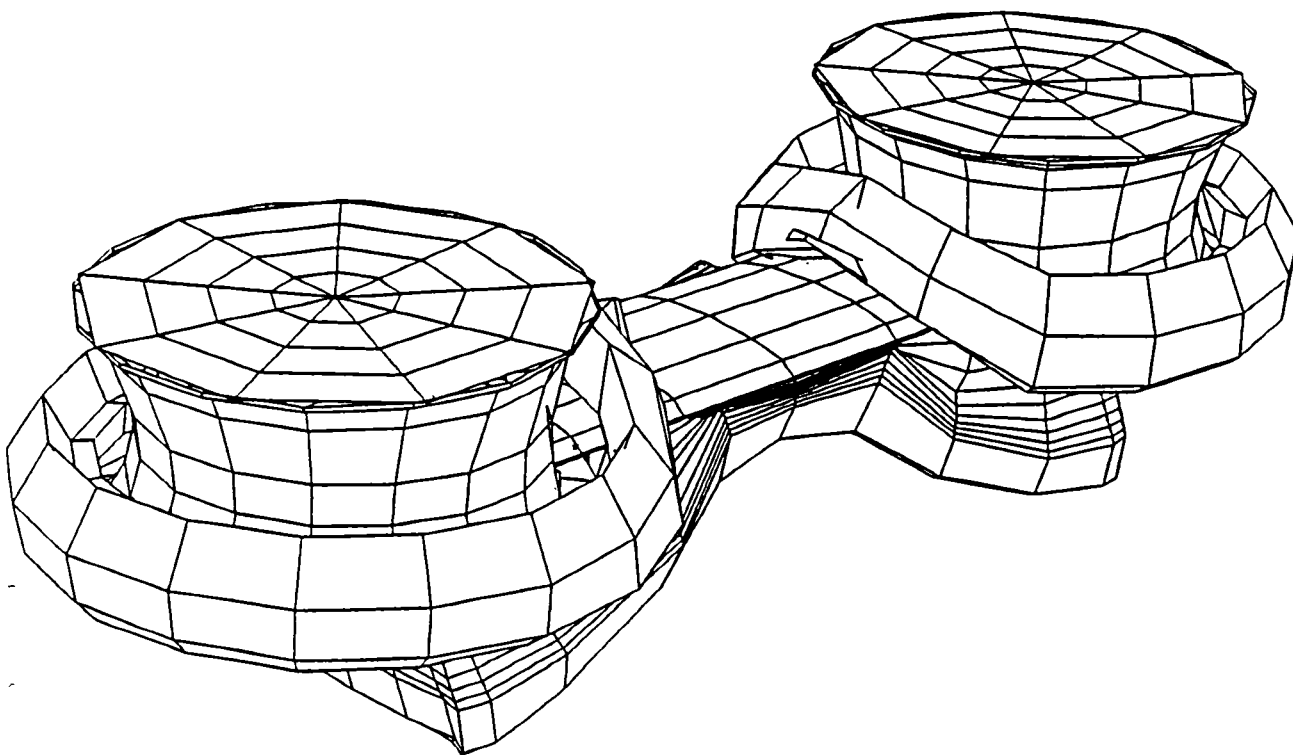
The download for a hovering tilt-rotor aircraft is typically 10 to 15% of the total rotor thrust. Since the payload of a tilt rotor is 25 to 30% of the gross weight, download reduction could mean 35 to 60% improvement on the payload. The complexity of the rotor and wing geometry and the flow features make the problem difficult to solve by a direct Navier-Stokes solver. A panel method can provide a relatively low-cost simulation without compromising too much on accuracy.

Future Plans

The three components of flow simulation are now being combined into a code capable of representing the hovering tilt-rotor configuration; the preliminary result is shown in the figure. After the code validation, the plan is to simulate the individual rotor blade instead of the actuator disk used in the present model. Since the memory required in this simulation is relatively small, the Cray Y-MP can support the development as well. Therefore, the NAS program was not applied for this year.

Publications

Lee, C. S. "Calculation of the Rotor Download on Airfoils." Presented at the OAST CFD Conference, NASA Ames Research Center, Mar. 1989.



Flow field of a hovering tilt rotor.

Simulation of the Climatology of the El Chichon Volcanic Aerosol Cloud in the Stratosphere

Richard E. Young, Principal Investigator

Co-investigators: Owen B. Toon and James B. Pollack

NASA Ames Research Center

Research Objective

The principal objective of this project is to numerically simulate the behavior of the El Chichon volcanic aerosol cloud in the stratosphere in order to better understand stratospheric transport and aerosol microphysical processes. An associated goal is to assess the climatic impact of the El Chichon volcanic eruption on stratospheric wind and temperature fields.

Approach

A three-dimensional, spectral, primitive-equation model is used to compute wind and temperature fields in the stratosphere. This model is coupled to a three-dimensional aerosol transport and microphysical model that computes the dispersion of the volcanic aerosol cloud using the computed winds from the circulation model, accounting for such processes as sedimentation and coagulation of the aerosol particles.

Accomplishment Description

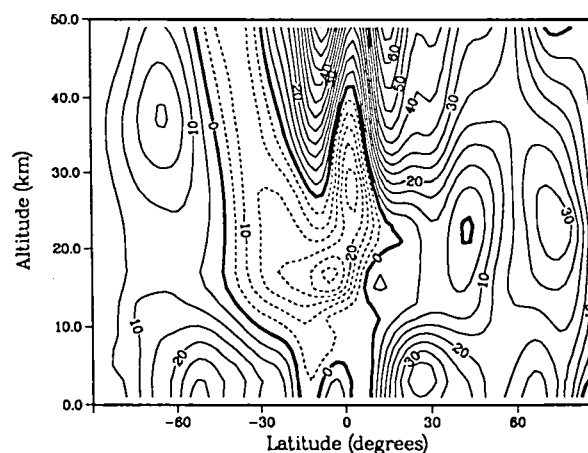
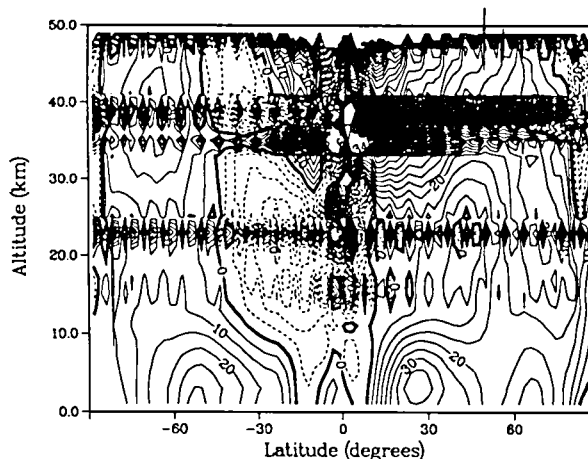
An improved balance equation was developed for initializing the three-dimensional circulation model wind and temperature fields. This effort was necessary because simulations of the stratosphere, typically needing about 12 Cray Y-MP hours and 10 megawords of memory, showed two anomalous features that were traced to dynamical imbalances in the model initial conditions. One anomaly was a short-wavelength oscillation in temperature as a function of latitude. The second was an unrealistic heating in the upper troposphere and lower stratosphere in the polar regions. We are still testing the new balance procedure to ensure that no other anomalous features are present that are caused by initial dynamical conditions. A diagnostic package to calculate and display the detailed energy and momentum balance of the three-dimensional circulation model was developed. This diagnostic package was instrumental in tracing the above-described anomalies to the initial conditions.

Significance

Major NASA programs, such as the Upper Atmospheric Research Program and the NASA Ames aircraft expeditions to the Antarctic in 1987 and the Arctic during the winter of 1988-1989, as well as various Earth-orbiting satellite programs such as the Upper Atmospheric Research Satellite program, have as principal goals the study of the climatology of the stratosphere. Further, the High Speed Research Program of OAET is currently sponsoring research to assess the climatic impact of high-speed aircraft operating in the lower stratosphere. A unique opportunity to better understand stratospheric transport and aerosol microphysical processes occurred when the El Chichon volcano erupted in April 1982, injecting some 10 million tons of aerosol particles into the stratosphere. A comprehensive climatology of the subsequent dispersal of the volcanic cloud, gathered from aircraft, satellite, and ground-based observations, has been compiled for the years immediately following the eruption. By simulating the behavior of the volcanic cloud we can assess and improve the theoretical understanding of transport and aerosol processes in the stratosphere, and thus make an important contribution to the above NASA programs.

Future Plans

Multi-year simulations of the undisturbed stratosphere will be carried out. The results from these calculations will be used to conduct passive tracer simulations of the El Chichon volcanic cloud. Once the passive tracer computations are complete, fully interactive tracer simulations will be done.



Height-latitude cross sections of longitudinally averaged east-west wind. Height is measured from the 300-mb pressure surface. The improvement resulting from the new balance procedure is evident in the bottom figure.

Development of an Euler-Based Method for Turboprop Integration

N. J. Yu, Principal Investigator
Co-investigators: H. C. Chen, T. J. Kao, and D. A. Naik
The Boeing Company/VIGYAN, Inc./NASA Langley Research Center

Research Objective

To develop an effective computational capability for the study of turboprop engine/airframe integration.

Approach

Transonic, three-dimensional, inviscid, multiblock, multigrid Euler codes were developed for the analysis of complete airplane configurations with either wing-mounted or aft-mounted propfans. A three-dimensional elliptic grid generation code that creates the volume grid from the specified surface grid was also developed. The propeller power effects were simulated using an actuator disc; either components of force and work distributions, or total pressure, total temperature, and swirl were prescribed along the disc as boundary conditions for the flow solver. A separate embedded flow solver provided detailed flow characteristics in the vicinity of the propulsive unit. This embedded solver inherited its starting values from the global solution.

Accomplishment Description

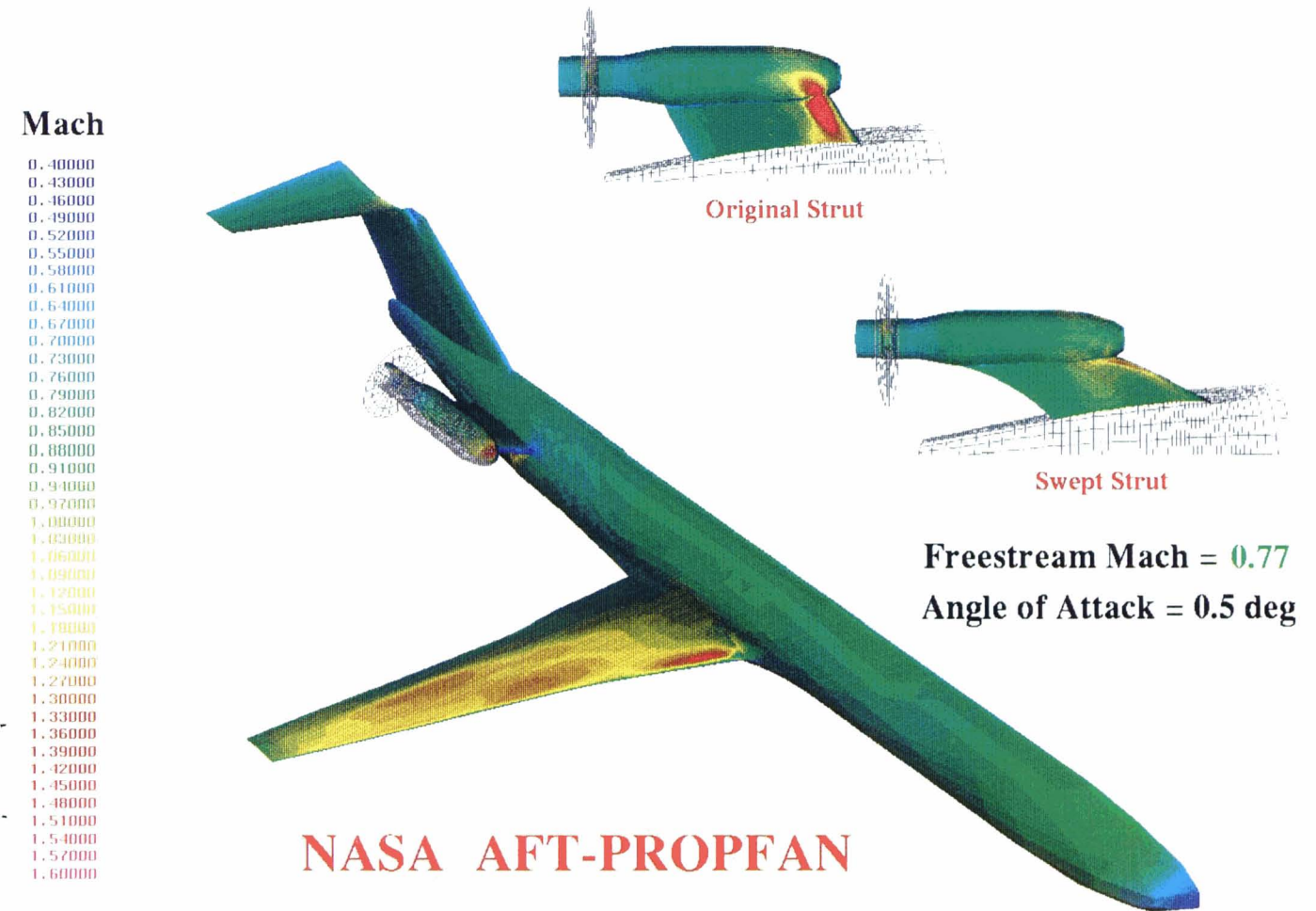
The code was successfully applied to the NASA aft-turboprop model shown in the accompanying figure. For this example, increasing the sweep of the strut was found to significantly reduce the adverse installation effects of propulsion integration. A typical analysis took about 2 hours on a Cray-2, used 295,000 grid points, and required 12 megawords of memory in the single-block mode or 3 megawords of memory in the multiblock mode. Embedded mesh solutions consumed less than 10 minutes of CPU time on the Cray-2.

Significance

The computational fluid dynamics (CFD) codes that were developed in this project have been used as an effective tool for propfan engine/airframe integration. Qualitative analysis of the CFD solutions have aided in the selection and modification of configurations for quantitative experimental analysis.

Future Plans

This experience is being applied to the development of a CFD code for turbofan/superfan installation.



Mach number contours for the NASA aft-mounted propfan; $M_\infty = 0.77$, $\alpha = 0.5^\circ$.

Incompressible Transition to Turbulence

T. A. Zang, Principal Investigator

Co-investigators: S. Dinavahi, B. Singer, and U. Piomelli
NASA Langley Research Center

Research Objective

The objective is to study the transition of fluid flows from smooth, laminar states to chaotic, turbulent states. The focus in 1989 was on wave interactions, on the subharmonic route to transition, and on modeling the final process of transition to turbulence.

Approach

Direct and large-eddy simulations of simple, low-speed channel and boundary-layer flows were performed under controlled conditions in the transitional regime. The simulations were performed for the time-dependent Navier-Stokes equations using highly accurate spectral methods.

Accomplishment Description

A weakly nonlinear theory was developed for predicting the effects of interaction between Görtler vortices and Tollmien-Schlichting waves in curved channel flow. Direct numerical simulations confirmed the validity of the theory. High-resolution direct simulations were performed of subharmonic transition in boundary layers. They revealed that laminar breakdown can occur on the large, primary scale or on smaller scales (see figure). Large-eddy simulations were conducted for both boundary-layer and channel-flow transition. They performed surprisingly well in predicting the peak skin friction in the transition zone. A total of 1540 CPU hours were used on the Cray-2 and Cray Y-MP during 1989-90.

Significance

The weakly nonlinear theory is a more economical predictive tool than numerical simulation, but the latter tool is essential

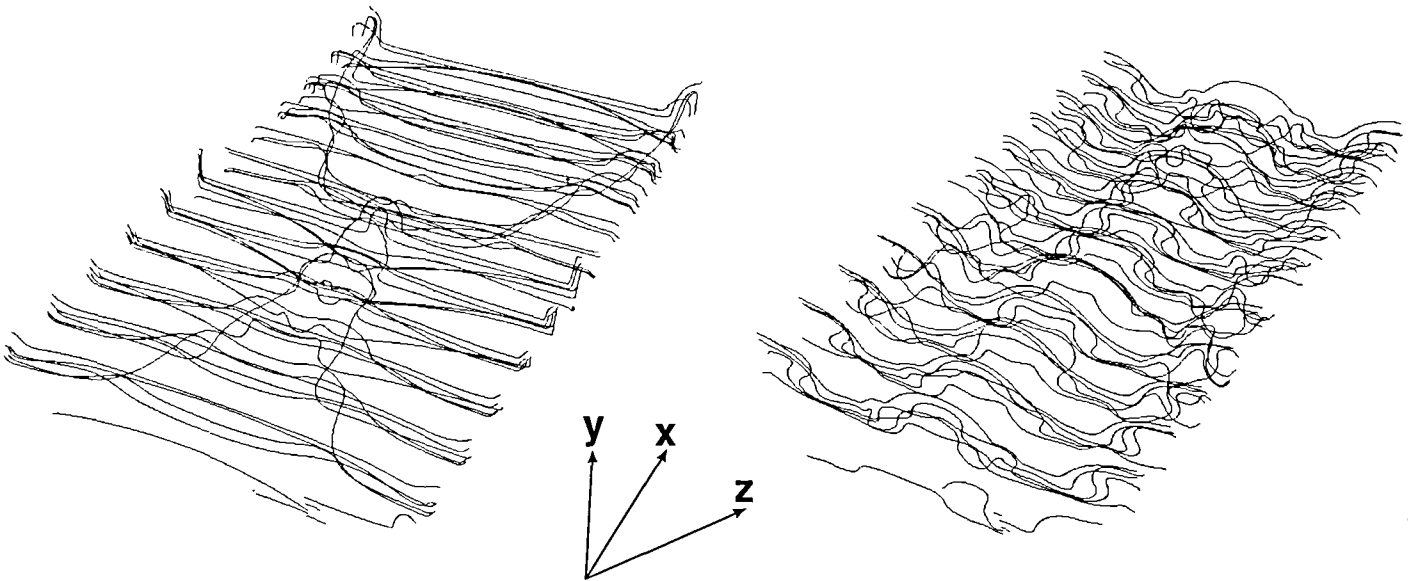
for corroborating the validity of this algebraically complex approach. The subharmonic transition simulations indicate the richness of the transition process, even in simple flows. Large-eddy simulations of transition are far less expensive than direct simulations, and the use of this new technology can lead to the expansion of the role of numerical computations of transition.

Future Plans

The detailed data bases for various subharmonic transition simulations will be thoroughly examined to clarify the different mechanisms at work. The large-eddy simulation method will be more thoroughly tested and then extended to spatial and compressible transition.

Publications

1. Singer, B. A.; Zang, T. A.; and Erlebacher, G. "TS-Dean Interactions in Curved Channel Flow." ICASE Report No. 90-9, 1990.
2. Zang, T. A., and Hussaini, M. Y. "Multiple Paths to Subharmonic Laminar Breakdown in a Boundary Layer." To be published in *Phys. Rev. Letters*.
3. Piomelli, U.; Zang, T. A.; Speziale, C. G.; and Hussaini, M. Y. "On the Large-Eddy Simulation of Transitional Wall-Bounded Flows." *Phys. Fluids A* 2 (1990): 257-265.
4. Zang, T. A.; Gilbert, N.; and Kleiser, L. "Direct Numerical Simulation of the Transitional Zone." *Proceedings of the NASA/ICASE Workshop on Instability and Transition*, ed. M. Y. Hussaini and R. G. Voight. New York: Springer, 1990.



Vortex lines showing laminar breakdowns in subharmonic transition.

Index by Research Sites

Allison Gas Turbine Division, General Motors Corporation

| | | |
|---------------------|--|-----|
| Edward J. Hall | <i>Ducted Propfan Analysis</i> | 59 |
| Richard H. Pletcher | <i>Development and Evaluation of Computational Methods for Turbomachinery Applications</i> (with Iowa State University) | 131 |
| Kurt F. Weber | <i>Simulation of Three-Dimensional Flow in a Transonic Fan Rotor</i> | 181 |

Analatom, Inc.

| | | |
|----------------------|---|-----|
| Robert W. MacCormack | <i>Simulation of Highly Ionized Flows</i> (with Stanford University) | 94 |
| Wolfgang Mueller | <i>Electronic Structure of Superconductors</i> | 120 |
| | (with NASA Lewis Research Center) | |

Analex Corporation

| | | |
|---------------|---|----|
| John L. Klann | <i>Solar Dynamic Space Power-System Design and Performance Analysis</i> | 80 |
| | (with NASA Lewis Research Center) | |

Analytical Services and Materials, Inc.

| | | |
|------------------------|--|----|
| William B. Compton III | <i>Transonic Navier-Stokes Solutions of Three-Dimensional Afterbody Flows</i> (with NASA Langley Research Center) | 24 |
| N. M. El-Hady | <i>Control of the Flow Field about an Airfoil Section by Localized Surface Heating</i> | 42 |
| | (with NASA Langley Research Center) | |
| L. Maestrello | <i>Radiation and Control of the Acoustic Field Caused by a Wave Packet Evolving along a Concave-Convex Surface</i> | 97 |
| | (with NASA Langley Research Center) | |

Bell Helicopter Textron, Inc.

| | | |
|------------------|--|-----|
| Matthew T. Scott | <i>Computational Methods for Rotor-Blade Drag Prediction</i> | 150 |
|------------------|--|-----|

Berkeley Research Associates

| | | |
|-------------------|--|-----|
| Stephen H. Brecht | <i>Hybrid Simulations of the Global Venus/Solar Wind Interaction</i> | 9 |
| Elaine S. Oran | <i>Unsteady Diffusion Flames</i> | 126 |
| | (with Naval Research Laboratory) | |

Boeing Advanced Systems

| | | |
|-------------------|--|----|
| Michael D. Madson | <i>Transonic Analysis About the F-16A and Other Complex Configurations</i> | 96 |
| | (with NASA Ames Research Center) | |

The Boeing Company

| | | |
|------------|--|-----|
| D. A. Naik | <i>Transonic Potential Flow about Transport Aircraft</i> (with ViGYAN Inc., NASA Langley Research Center) | 121 |
| N. J. Yu | <i>Development of an Euler-Based Method for Turboprop Integration</i> | 191 |
| | (with NASA Langley Research Center, ViGYAN, Inc.) | |

California Institute of Technology

| | | |
|--------------------|--|-----|
| Gregory A. Lyzenga | <i>Earthquake-Cycle Stress Simulations</i> (with Jet Propulsion Laboratory) | 91 |
| Robert A. West | <i>Optical Properties of Nonspherical Particles in Planetary Atmospheres</i> | 183 |
| | (with Jet Propulsion Laboratory) | |
| Richard G. Wilmoth | <i>Shock Interactions in Rarefied Hypersonic Flows</i> | 185 |
| | (with NASA Langley Research Center) | |

Case Western Reserve University

| | | |
|----------------|---|----|
| Steven H. Izen | <i>Numerical Inversion of a Limited-Data X-Ray Transform.....</i> | 78 |
|----------------|---|----|

Clarkson University

| | | |
|------------------------|--|----|
| Nabeel A. O. Demerdash | <i>Finite-Element Computation of Three-Dimensional Magnetic Fields for Design Optimization of Generators for Space Station Solar Dynamic Power</i> | 30 |
|------------------------|--|----|

Colorado State University

| | | |
|------------------|--|-----|
| David A. Randall | <i>Modeling Global Cloudiness.....</i> | 140 |
|------------------|--|-----|

David Taylor Research Center

| | | |
|--------------------|---|-----|
| Henry J. Haussling | <i>Multiblock Solutions of the Navier-Stokes Equations</i> | 67 |
| Thomas T. Huang | <i>Turbulence Modeling for Submarine Flow Field Computation</i> | 73 |
| Chao-Ho Sung | <i>Naval Applications of Computational Fluid Dynamics</i> | 165 |
| Tsze C. Tai | <i>Hydrodynamic Prediction Methods for Advanced Submarine Configurations.....</i> | 167 |
| Tsze C. Tai | <i>Low-Speed Maneuver Aerodynamics in a Nonuniform Free Stream</i> | 168 |
| Cheng-I Yang | <i>Numerical Simulation of Submarine Propulsion.....</i> | 187 |

Douglas Aircraft Company

| | | |
|-------------|--|----|
| Lee T. Chen | <i>Calculation of Transonic and Supersonic Flows Using an Interactive Scheme Based on Euler and Boundary-Layer Equations</i> | 21 |
|-------------|--|----|

General Dynamics, Convair Division

| | | |
|-----------------|--|-----|
| Adam C. Bricker | <i>Hypersonic Air-Breathing Missile Aero/Propulsion Integration.....</i> | 11 |
| Geoffrey Butler | <i>Two-Body Hypersonic Separation Analysis</i> | 14 |
| Peter K. Shih | <i>Afterbody Aerothermodynamic Studies for the Hypersonic Glide Vehicle.....</i> | 153 |

General Dynamics, Fort Worth Division

| | | |
|-------------------|--|----|
| Robert DeParvine | <i>High-Angle-of-Attack Inlet Analysis and Design Using Computational Fluid Dynamics Methods</i> | 31 |
| Robert DeParvine | <i>Validation of a Computational Fluid Dynamics Code for High-Speed-Inlet Design</i> | 32 |
| Richard L. Haller | <i>Evaluation of the CAP-TSD Computer Code Using an F-16 Study Case</i> | 60 |

General Electric Aircraft Engines

| | | |
|--------------------------|--|-----|
| Sundaresa V. Subramanian | <i>Numerical Study of Three-Dimensional Viscous Reacting Flows with Application to Scramjet Propulsion</i> | 163 |
|--------------------------|--|-----|

High Technology Corporation

| | | |
|-----------------|--|----|
| Mujeeb R. Malik | <i>Transition in Hypersonic and Three-Dimensional Boundary Layers</i> (with NASA Langley Research Center) | 98 |
|-----------------|--|----|

ICASE

| | | |
|----------------------|--|-----|
| G. Erlebacher | <i>Compressible Turbulence.....</i> (with NASA Langley Research Center) | 44 |
| Dimitri J. Mavriplis | <i>Solution of the Steady State Navier-Stokes Equations on Unstructured and Adaptive Meshes</i> (with NASA Langley Research Center) | 102 |

Iowa Institute of Hydraulic Research

| | | |
|-------------|--|-----|
| V. C. Patel | <i>Hydrodynamics of Self-Propelled Bodies.....</i> | 128 |
|-------------|--|-----|

Iowa State University

| | | |
|---------------------|---|-----|
| James C. Hill | <i>Direct Simulation of Turbulent Reacting Flows</i> | 69 |
| Richard H. Pletcher | <i>Development and Evaluation of Computational Methods for Turbomachinery Applications</i> (with Allison Gas Turbine Division, General Motors Corporation) | 131 |
| John C. Tannehill | <i>Development of a Robust Parabolized Navier-Stokes Code for Computing Three-Dimensional, Chemically Reacting Flow Fields</i> | 171 |

JAYCOR

| | | |
|--------------------|--|----|
| Harley E. Hurlburt | <i>Eddy-Resolving Model of the Pacific Ocean</i> (with the Naval Oceanographic and Atmospheric Research Laboratory) | 76 |
|--------------------|--|----|

Jet Propulsion Laboratory

| | | |
|--------------------|---|-----|
| Gregory A. Lyzenga | <i>Earthquake-Cycle Stress Simulations</i> (with California Institute of Technology) | 91 |
| Robert A. West | <i>Optical Properties of Nonspherical Particles in Planetary Atmospheres</i> (with California Institute of Technology) | 183 |

Kansas State University

| | | |
|---------------|---|----|
| Kadosa Halasi | <i>Parameter Study in the Driven Cavity</i> (with University of Colorado at Boulder) | 58 |
|---------------|---|----|

Lockheed Aeronautical Systems Company

| | | |
|-------------------|---|-----|
| Charles R. Olling | <i>Navier-Stokes Simulation of Separated Turbulent Flow around a Generic Fighter Aircraft</i> | 124 |
| Charles R. Olling | <i>Navier-Stokes Simulation of Turbulent Flow over Rectangular Cavities Containing Stores</i> | 125 |

Lockheed Engineering and Sciences Company

| | | |
|-------------------|--|-----|
| Robert A. Golub | <i>Flow-Field Simulation Around Different Rotor Planforms</i> (with NASA Langley Research Center) | 52 |
| Edward C. Ma | <i>Numerical Investigation of a Space Shuttle Orbiter Contingency Abort</i> | 92 |
| Ajay K. Pandey | <i>Flux-Based Finite-Element Formulation Reduces CPU Time for Thermal-Structural Analysis</i> (with NASA Langley Research Center) | 127 |
| Ramadas K. Prabhu | <i>Finite-Element Analysis of Equilibrium and Nonequilibrium Flows</i> (with NASA Langley Research Center) | 133 |
| Luen T. Tam | <i>Thermochemical and Radiative Nonequilibrium Flow Simulation for the Aeroassist Flight Experiment</i> | 170 |
| Thomas C. Wey | <i>Application of Unstructured Grids to Shuttle Computational Fluid Dynamics</i> (with NASA Johnson Space Center) | 184 |

Lockheed Missiles and Space Company, Inc.

| | | |
|------------------------|---|----|
| R. Rexford Chamberlain | <i>Three-Dimensional Calculations of Multiple Control-Jet Interactions for Defense Interceptors</i> | 18 |
|------------------------|---|----|

Los Alamos National Laboratory

| | | |
|--------------------|---|----|
| Gary A. Glatzmaier | <i>Numerical Simulations of Deep Global Convection in Jupiter</i> | 50 |
|--------------------|---|----|

McDonnell Aircraft Company

| | | |
|-------------------|--|----|
| Shreekant Agrawal | <i>Separated and Vortical Flow-Field Analysis on Fighter Wing Configurations</i> | 4 |
| Raymond R. Cosner | <i>Hornet 2000 Flow-Field Analysis</i> | 25 |
| John Mangus | <i>Navier-Stokes Analysis of the Air Force Vortex Flap Model and the F/A-18</i> | 99 |
| | (with Northrop Aircraft Division) | |

| | | |
|--|--|-----|
| James A. Rhodes | <i>Computation of ASTOVL Flow Fields</i> | 141 |
| McDonnell Douglas Corporation | | |
| C. C. Lee | <i>Numerical Investigation of NASP-Like Vehicles</i> | 85 |
| McDonnell Douglas Research Laboratories | | |
| Ramesh K. Agarwal | <i>Helicopter Fuselage and Rotor Flow-Field Calculations in Hover and Forward Flight</i> (with NASA Ames Research Center) | 2 |
| Ramesh K. Agarwal | <i>Numerical Solution of the Reynolds-Averaged Navier-Stokes Equations for Flow About an Almost Compete Aircraft</i> (with NASA Ames Research Center) | 3 |
| William W. Bower | <i>Calculations of Unsteady, Supersonic-Jet Flow Fields and Acoustics</i> (with Nielsen Engineering & Research, Inc.) | 8 |
| Alan B. Cain | <i>Receptivity, Eigenfunction Modeling, and Simulation of a Wall-Bounded Flow</i> | 15 |
| NASA Ames Research Center | | |
| Ramesh K. Agarwal | <i>Helicopter Fuselage and Rotor Flow-Field Calculations in Hover and Forward Flight</i> (with McDonnell Douglas Research Laboratories) | 2 |
| Ramesh K. Agarwal | <i>Numerical Solution of the Reynolds-Averaged Navier-Stokes Equations for Flow About an Almost Compete Aircraft</i> (with McDonnell Douglas Research Laboratories) | 3 |
| Donald Baganoff | <i>Discrete Particle Simulation of Compressible Flow</i> (with Stanford University) | 5 |
| Jeffrey C. Buell | <i>Three-Dimensional Shear Layers and Wall-Bounded Compressible Turbulence</i> | 12 |
| Pieter G. Buning | <i>Space Shuttle Flow Field</i> | 13 |
| Alan B. Cain | <i>Receptivity, Eigenfunction Modeling, and Simulation of a Wall-Bounded Flow</i> | 15 |
| Denny S. Chaussee | <i>Hypersonic Flow Past Generic Lifting Bodies</i> | 20 |
| W. J. Chyu | <i>Airframe/Inlet Aerodynamics</i> | 22 |
| | NASA Ames Research Center | |
| Bill Davy | <i>Three-Dimensional Aeroassist Flight Experiment Flow Simulations</i> | 28 |
| Thomas A. Edwards | <i>Hypersonic Chemically Reacting Flow over Blended Wing-Body Vehicles</i> | 37 |
| Thomas A. Edwards | <i>Numerical Simulation of Hydrogen-Air Combustion in Hypersonic-Vehicle Propulsion Systems</i> | 38 |
| Fort F. Felker | <i>Three-Dimensional Viscous Drag Prediction for Rotor Blades</i> | 45 |
| Jolen Flores | <i>Compressible Navier-Stokes Code Development</i> | 46 |
| Aga M. Goodsell | <i>A Supersonic Computational Fluid Dynamic Analysis of a Generic Fighter</i> | 53 |
| G. P. Guruswamy | <i>Simulation of Fluid/Structural Interactions</i> | 56 |
| C. C. Horstman | <i>Turbulence Modeling for Compressible/Hypersonic Flows</i> | 71 |
| Ching-mao Hung | <i>Computation of Steady Three-Dimensional Separation</i> | 75 |
| Laurence R. Keefe | <i>Linking Attractor Geometry to Turbulence Physics</i> | 79 |
| Dochan Kwak | <i>Incompressible Navier-Stokes Calculations for the Space Shuttle Main Engine</i> | 83 |

| | | |
|---|--|-----|
| Robert D. MacElroy | <i>Computer Simulation of Ion Transport across Membranes</i> | 95 |
| Michael D. Madson | <i>Transonic Analysis About the F-16A and Other Complex Configurations</i> (with Boeing Advanced Systems) | 96 |
| Nagi N. Mansour | <i>Direct Simulation of Compressible Turbulent Flows</i> | 100 |
| Fred W. Martin, Jr. | <i>Space Shuttle Flow Field</i> (with NASA Johnson Space Flight Center) | 101 |
| W. J. McCroskey | <i>Aerodynamic Flows about High-Performance Rotor Blade Tips</i> (with U. S. Army Aeroflightdynamics Directorate—AVSCOM) | 105 |
| W. J. McCroskey | <i>Airloads and Acoustics of Rotorcraft</i> (with U. S. Army Aeroflightdynamics Directorate—AVSCOM) | 106 |
| W. J. McCroskey | <i>Tilt-Rotor Aerodynamic Interactions</i> | 107 |
| John E. Melton | <i>Computational Fluid Dynamics Analysis of Advanced Turboprop Configurations</i> | 110 |
| Gene P. Menees | <i>Standing Oblique Detonation Waves</i> | 112 |
| Robert D. Moser | <i>Wall-Bounded Turbulent Flows</i> | 119 |
| Thomas H. Pulliam | <i>Chaotic, Unsteady, Low-Reynolds Number Navier-Stokes Flow</i> | 134 |
| Man Mohan Rai | <i>A Finite-Difference Approach to Direct Simulations and Large-Eddy Simulations of Turbulent Flow</i> | 135 |
| Man Mohan Rai | <i>Three-Dimensional Multi-Airfoil Navier-Stokes Simulations of Rotor/Stator Interaction in Turbines</i> | 136 |
| Man Mohan Rai | <i>Unsteady Flow in a Multistage Compressor</i> | 137 |
| Michael M. Rogers | <i>Turbulent Reacting Flows</i> | 144 |
| William C. Rose | <i>National Aero-Space Plane Inlet Flow Fields</i> (with Rose Engineering & Research, Inc.) | 147 |
| William C. Rose | <i>Three-Dimensional Viscous Flow in High-Speed Inlets</i> (with Rose Engineering & Research, Inc.) | 148 |
| Philippe R. Spalart | <i>Transition and Turbulence in Three-Dimensional Flows</i> | 156 |
| Roger C. Strawn | <i>Computational Fluid Dynamics Prediction of Advanced Rotor Performance</i> (with U.S. Army Aeroflightdynamics Directorate—AVSCOM) | 159 |
| Larry Young | <i>Tilt-Rotor Download Prediction</i> | 189 |
| Richard E. Young | <i>Simulation of the Climatology of the El Chichon Volcanic Aerosol Cloud in the Stratosphere</i> | 190 |
| NASA Goddard Space Flight Center | | |
| David Adamec | <i>Large-Scale Ocean Modeling</i> | 1 |
| Hans G. Mayr | <i>Numerical Study of Planetary Atmospheres</i> | 103 |
| David A. Randall | <i>Modeling Global Cloudiness</i> | 140 |
| NASA Johnson Space Center | | |
| Fred W. Martin, Jr. | <i>Space Shuttle Flow Field</i> (with NASA Ames Research Center) | 101 |
| Phil C. Stuart | <i>Mars Rover/Sample Return Aerocapture Vehicle</i> | 162 |

| | |
|-------------------------------------|---|
| Luen T. Tam | <i>Thermochemical and Radiative Nonequilibrium Flow Simulation for the Aeroassist Flight Experiment</i> 170 (with Lockheed Engineering and Sciences Company) |
| Thomas C. Wey | <i>Application of Unstructured Grids to Shuttle Computational Fluid Dynamics</i> 184 (with Lockheed Engineering and Sciences Company) |
| NASA Langley Research Center | |
| John T. Batina | <i>Validation of a Three-Dimensional Flux-Split Euler Algorithm for Unstructured Grids</i> 7 |
| William B. Compton III | <i>Transonic Navier-Stokes Solutions of Three-Dimensional Afterbody Flows</i> 24 (with Analytical Services and Materials, Inc.) |
| Pramote Dechaumphai | <i>Integrated Fluid-Thermal-Structural Analyzer Demonstrates Heat-Transfer/Deformation Coupling</i> 29 |
| J. Philip Drummond | <i>Simulation of Supersonic Chemically Reacting Flow Fields</i> 34 |
| N. M. El-Hady | <i>Control of the Flow Field about an Airfoil Section by Localized Surface Heating</i> 42 (with Analytical Services and Materials, Inc.) |
| G. Erlebacher | <i>Compressible Turbulence</i> 44 (with ICASE) |
| Javier A. Garriz | <i>Validation of 3-D Wind Tunnel Wall Interference Assessment/Correction Codes</i> 49 |
| Peter A. Gnoffo | <i>Hypersonic Flows in Chemical and Thermal Nonequilibrium</i> 51 |
| Robert A. Golub | <i>Flow-Field Simulation Around Different Rotor Planforms</i> 52 (with Lockheed Engineering and Sciences Company) |
| W. L. Grose | <i>Three-Dimensional Atmospheric Simulation Model</i> 55 |
| Julius E. Harris | <i>Supersonic Laminar-Flow-Control Computational Fluid Dynamics</i> 61 |
| Peter M. Hartwich | <i>General, Fast, Accurate Floating Shock Fitting Procedure for Unadapted Meshes</i> 62 (with ViGYAN, Inc.) |
| Peter M. Hartwich | <i>Navier-Stokes Solutions for Vortical Flows over a Forebody</i> 63 (with ViGYAN, Inc.) |
| C.-H. Hsu | <i>Prediction of Vortical Flows Using Incompressible Navier-Stokes Equations</i> 72 (with ViGYAN, Inc.) |
| Ajay Kumar | <i>Three-Dimensional Shock/Shock Interactions on the Inlet Side Wall</i> 81 |
| Geojoe Kuruvila | <i>Numerical Study of Fundamental Fluid Dynamics using Navier-Stokes Equations</i> 82 |
| C. H. Liu | <i>Computational Prediction and Control of Steady and Unsteady Asymmetric Vortex Flows around Cones</i> 89 (with Old Dominion University) |
| James M. Luckring | <i>Transonic Navier-Stokes Solutions about a Complex High-Speed Accelerator Configuration</i> 90 |
| M. G. Macaraeg | <i>Nonlinear Evolution of Supersonic Disturbances in Mixing Layers</i> 93 |
| L. Maestrello | <i>Radiation and Control of the Acoustic Field Caused by a Wave Packet Evolving along a Concave-Convex Surface</i> 97 (with Analytical Services and Materials, Inc.) |
| Mujeeb R. Malik | <i>Transition in Hypersonic and Three-Dimensional Boundary Layers</i> 98 (with High Technology Corporation) |

| | | |
|-----------------------------------|--|-----|
| Dimitri J. Mavriplis | <i>Solution of the Steady State Navier-Stokes Equations on Unstructured and Adaptive Meshes</i> (with ICASE) | 102 |
| Charles R. McClinton | <i>National Aero-Space Plane Propulsion Flow-Path Analysis and Code Certification</i> | 104 |
| S. Naomi McMillin | <i>Incipient Leading-Edge Separation</i> | 108 |
| N. Duane Melson | <i>Multiblock, Multigrid Method for the Solution of the Three-Dimensional Euler Equations</i> | 109 |
| Joseph H. Morrison | <i>Multi-Tasked Numerical Simulation of Complex Configurations in Hypersonic Flow</i> | 118 |
| D. A. Naik | <i>Transonic Potential Flow about Transport Aircraft</i> (with ViGYAN Inc., The Boeing Company) | 121 |
| Ajay K. Pandey | <i>Flux-Based Finite-Element Formulation Reduces CPU Time for Thermal-Structural Analysis</i> (with Lockheed Engineering and Sciences Co.) | 127 |
| James L. Pittman | <i>Generic National Aero-Space Plane Afterbody</i> | 129 |
| James L. Pittman | <i>NASP-TMP Forebody/Inlet Integration</i> | 130 |
| Ramadas K. Prabhu | <i>Finite-Element Analysis of Equilibrium and Nonequilibrium Flows</i> (with Lockheed Engineering & Sciences Co.) | 133 |
| R. Ramakrishnan | <i>An Adaptation Procedure Combining Mesh Refinement with Mesh Movement for Compressible Flows</i> | 139 |
| R. Clayton Rogers | <i>Hydrogen-Air Combustion Simulation in a Pulse Facility</i> | 145 |
| Balu Sekar | <i>Direct Simulation of High-Speed Mixing Layers With and Without Chemical Heat Release</i> | 151 |
| John V. Shebalin | <i>Ionized Flow Around a Reentry Vehicle</i> | 152 |
| Craig L. Streett | <i>Numerical Simulation of Wave Interactions Related to Transition</i> | 160 |
| Craig L. Streett | <i>Transition Simulations in Nonhomogeneous Geometries</i> | 161 |
| R. C. Swanson | <i>Development of Algorithms for Solving the Three-Dimensional Navier-Stokes Equations</i> | 166 |
| Rajiv Thareja | <i>Computation of Three-Dimensional Flows Using Unstructured Grids</i> | 172 |
| Rajiv Thareja | <i>Viscous Hypersonic Flow over a 24-Degree Compression Corner</i> | 173 |
| Veer N. Vatsa | <i>High Reynolds Number, Transonic, Viscous Flow over Aircraft Components</i> | 177 |
| Kenneth J. Weilmuenster | <i>Winged Entry Vehicle Computations</i> | 182 |
| Richard G. Wilmoth | <i>Shock Interactions in Rarefied Hypersonic Flows</i> | 185 |
| | (with California Institute of Technology) | |
| N. J. Yu | <i>Development of an Euler-Based Method for Turboprop Integration</i> | 191 |
| | (with The Boeing Company, ViGYAN, Inc.) | |
| T. A. Zang | <i>Incompressible Transition to Turbulence</i> | 192 |
| NASA Lewis Research Center | | |
| William J. Coirier | <i>Three-Dimensional Shock/Boundary-Layer Interactions in Hypersonic Inlets</i> | 23 |
| James D. Heidmann | <i>An Analysis of the Viscous Flow through a Compact Radial Turbine by the Average-Passage Approach</i> | 68 |
| Danny Hwang | <i>Computational Advanced Propulsion Technology Research</i> | 77 |
| | | 199 |

| | | |
|--|--|-----|
| John L. Klann | <i>Solar Dynamic Space Power-System Design and Performance Analysis</i> (with Analex Corporation) | 80 |
| S. J. Leib | <i>Nonlinear Instability of Shear Layers</i> (with Sverdrup Technology, Inc.) | 86 |
| Christopher J. Miller | <i>Numerical Simulation of Advanced Propellers</i> (with Sverdrup Technology, Inc.) | 116 |
| Wolfgang Mueller | <i>Electronic Structure of Superconductors</i> (with Analatom, Inc.) | 120 |
| R. M. Nallasamy | <i>Unsteady Flow Field on an Advanced Propeller</i> (with Sverdrup Technology, Inc.) | 122 |
| M. S. Raju | <i>Analysis of Rotary Engine Processes and Computer Codes</i> (with Sverdrup Technology, Inc.) | 138 |
| William C. Rose | <i>National Aero-Space Plane Inlet Boundary Layer Control, Phase II</i> (with Rose Engineering & Research, Inc.) | 146 |
| Jian-Shun Shuen | <i>Advanced Numerical Algorithms for Chemically Reacting Flows</i> (with Sverdrup Technology, Inc.) | 154 |
| B. M. Steinetz | <i>National Aero-Space Plane Engine Corner-Flow</i> (with Sverdrup Technology, Inc.) | 158 |
| NASA Marshall Space Flight Center | | |
| Edwin B. Brewer | <i>Direct Simulation of Aerothermal Loads for the Aeroassist Flight Experiment</i> | 10 |
| Naval Oceanographic and Atmospheric Research Laboratory | | |
| Harley E. Hurlburt | <i>Eddy-Resolving Model of the Pacific Ocean</i> (with JAYCOR) | 76 |
| Naval Research Laboratory | | |
| Russell B. Dahlburg | <i>Dynamical Modeling of the Solar Atmosphere</i> | 26 |
| Fernando F. Grinstein | <i>Simulation of Compressible, Spatially Evolving, Reactive Planar Shear Flows</i> | 54 |
| Elaine S. Oran | <i>Unsteady Diffusion Flames</i> (with Berkeley Research Associates) | 126 |
| Nielsen Engineering & Research, Inc. | | |
| William W. Bower | <i>Calculations of Unsteady, Supersonic-Jet Flow Fields and Acoustics</i> (with McDonnell Douglas Research Laboratories) | 8 |
| Steven C. Caruso | <i>An Unstructured Triangular-Mesh/Navier-Stokes Method for Computing the Aerodynamics of Aircraft with Ice Accretion</i> | 16 |
| Michael R. Mendenhall | <i>Vortex-Induced Nonlinearities on Submarines</i> | 111 |
| David Nixon | <i>Pegasus™ Aerodynamic Analyses</i> | 123 |
| North Carolina State University | | |
| H. A. Hassan | <i>Monte Carlo Simulation of Hypersonic Flows</i> | 64 |
| H. A. Hassan | <i>Turbulence Modeling of Separated Flows</i> | 65 |
| H. A. Hassan | <i>Turbulent Supersonic Mixing Layers</i> | 66 |
| Scott D. Holland | <i>A Computational and Experimental Parametric Study of Three-Dimensional Side-Wall Compression Scramjet Inlets at Mach 10</i> | 70 |

Northrop Aircraft Division

| | | |
|--------------|--|----|
| Rung T. Ling | <i>Application of Computational Fluid Dynamics Methods to Radar Cross Section Computations</i> | 88 |
| John Mangus | <i>Navier-Stokes Analysis of the Air Force Vortex Flap Model and the F/A-18</i> (with McDonnell Aircraft Company) | 99 |

Northwestern University

| | | |
|-------------------|---|----|
| Arthur J. Freeman | <i>Electronic-Structure Study of High Superconductors</i> | 47 |
|-------------------|---|----|

Old Dominion University

| | | |
|-----------|--|----|
| C. H. Liu | <i>Computational Prediction and Control of Steady and Unsteady Asymmetric Vortex Flows around Cones</i> (with NASA Langley Research Center) | 89 |
|-----------|--|----|

The Pennsylvania State University

| | | |
|-----------------------|---|-----|
| George S. Dulikravich | <i>Acceleration of Iterative Algorithms for Euler and Navier-Stokes Equations</i> | 35 |
| Charles L. Merkle | <i>Absorption of Microwave Energy in a Flowing Gas</i> | 115 |

QUEST Integrated, Inc.

| | | |
|--------------|--|-----|
| Suresh Menon | <i>Large-Eddy Simulations of Ramjet Combustion Instability</i> | 113 |
| Suresh Menon | <i>Mixing Enhancement for Scramjet Flameholders</i> | 114 |

Rockwell International, North American Aircraft Division

| | | |
|----------------|--|-----|
| Jong H. Wang | <i>Navier-Stokes Predictions of Hypersonic Chemically Reacting Flows</i> | 180 |
| Chung-Jin Woan | <i>Computational Fluid Dynamics Design of a Laminar-Flow Control Glove</i> | 186 |

Rockwell International, Rocketdyne Division

| | | |
|---------------|--|-----|
| S. J. Lin | <i>Computational Fluid Dynamics Analysis of Space Shuttle Main Engine Turbopump Rotor/Stator Flows in Two and Three Dimensions</i> | 87 |
| Ruey-Jen Yang | <i>Multistage Rotor/Stator Interaction in the Space Shuttle Main Engine Turbopump Turbine</i> | 188 |

Rockwell International Science Center

| | | |
|-------------------------|--|----|
| Sukumar R. Chakravarthy | <i>Fuel/Air Turbulent Combustion in a Three-Dimensional Combustor Geometry in Supersonic Flow</i> | 17 |
| Dan F. Dominik | <i>Advanced Solid Rocket Motor Study</i> (with Rockwell International, Space Transportation Systems Division) | 33 |

Rockwell International, Space Transportation Systems Division

| | | |
|----------------|--|----|
| Dan F. Dominik | <i>Advanced Solid Rocket Motor Study</i> (Rockwell International, Science Center) | 33 |
|----------------|--|----|

Rose Engineering & Research, Inc.

| | | |
|-----------------|--|-----|
| William C. Rose | <i>National Aero-Space Plane Inlet Boundary Layer Control, Phase II</i> (with NASA Lewis Research Center) | 146 |
| William C. Rose | <i>National Aero-Space Plane Inlet Flow Fields</i> (with NASA Ames Research Center) | 147 |
| William C. Rose | <i>Three-Dimensional Viscous Flow in High-Speed Inlets</i> (with NASA Ames Research Center) | 148 |

Spectrex Inc.

| | | |
|--------------------|--|---|
| R. Balasubramanian | <i>Direct Simulation of Large-Eddy Breakup Devices</i> | 6 |
|--------------------|--|---|

Stanford University

| | | |
|-----------------|--|---|
| Donald Baganoff | <i>Discrete Particle Simulation of Compressible Flow</i> (with NASA Ames Research Center) | 5 |
|-----------------|--|---|

| | | |
|---|--|-----|
| Dean R. Chapman | <i>Continuum Computational Fluid Dynamics for Hypersonic High-Altitude Flight</i> | 19 |
| Professor John K. Eaton | <i>The Interaction of Particles with Homogeneous Turbulence</i> | 36 |
| Robert W. McCormack | <i>Simulation of Highly Ionized Flows</i> (with Analatom, Inc.) | 94 |
| Stanford University School of Medicine | | |
| Dennis Sullivan | <i>Microwave Hyperthermia Computer Modeling</i> | 164 |
| Sverdrup Technology, Inc. | | |
| H. T. Lai | <i>Computation of Single-Expansion-Ramp and Scramjet Nozzles</i> | 84 |
| S. J. Leib | <i>Nonlinear Instability of Shear Layers</i> (with NASA Lewis Research Center) | 86 |
| Christopher J. Miller | <i>Numerical Simulation of Advanced Propellers</i> (with NASA Lewis Research Center) | 116 |
| R. M. Nallasamy | <i>Unsteady Flow Field on an Advanced Propeller</i> (with NASA Lewis Research Center) | 122 |
| M. S. Raju | <i>Analysis of Rotary Engine Processes and Computer Codes</i> (with NASA Lewis Research Center) | 138 |
| Jian-Shun Shuen | <i>Advanced Numerical Algorithms for Chemically Reacting Flows</i> (with NASA Lewis Research Center) | 154 |
| B. M. Steinetz | <i>National Aero-Space Plane Engine Corner-Flow</i> (with NASA Lewis Research Center) | 158 |
| United Technologies, Pratt & Whitney | | |
| John J. Erhart | <i>CFD Analysis of the Flow Paths of Spherical Convergent Flap Nozzles</i> | 43 |
| Ronald K. Takahashi | <i>Unsteady Euler Analysis of the Redistribution of an Inlet Temperature Distortion in a Turbine</i> | 169 |
| United Technologies Research Center | | |
| Roger L. Davis | <i>The Impact of Hot-Streak Migration on Turbine Heat Transfer</i> | 27 |
| T. Alan Egolf | <i>Advanced CFD Codes for Rotary-Wing Airloads and Performance Prediction</i> | 39 |
| University of Arizona | | |
| K.-Y. Fung | <i>Computational Studies of Compressibility Effects on Dynamic Stall</i> | 48 |
| University of California, Berkeley | | |
| William H. Miller | <i>Quantum Mechanical Reactive Scattering</i> | 117 |
| Andrew Pohorille | <i>Vapor-Phase Growth of Crystals in Microgravity</i> | 132 |
| University of California, Irvine | | |
| S. E. Elghobashi | <i>Direct Simulation of Turbulent Reacting Flows</i> | 40 |
| S. E. Elghobashi | <i>Dispersion of Solid and Fluid Particles in Turbulent Homogeneous Flows With and Without Turbulence Modulation</i> | 41 |
| William A. Sirignano | <i>Ignition and Flame Spread above Liquid Fuel Pools: Gravity Effects</i> | 155 |
| University of California, Santa Barbara | | |
| Frank J. Spera | <i>Magma Chamber Convection</i> | 157 |
| University of Colorado, Boulder | | |
| Karl E. Gustafson | <i>Vortex Dynamics in Aerodynamic Flows</i> | 57 |

| | | |
|--|--|-----|
| Kadosa Halasi | <i>Parameter Study in the Driven Cavity.....</i> | 58 |
| | (with Kansas State University) | |
| University of Evansville | | |
| Ben R. Riley | <i>Kinetic Theory Model for the Flow of a Simple Gas from a Three-Dimensional Axisymmetric Nozzle.....</i> | 142 |
| University of Illinois, Urbana-Champaign | | |
| M. B. Salamon | <i>Advanced Computational Materials</i> | 149 |
| Pratap Vanka | <i>Hot Gas Ingestion by STOVL Aircraft.....</i> | 175 |
| University of Iowa | | |
| V. C. Patel | <i>Hydrodynamics of Self-Propelled Bodies.....</i> | 128 |
| | (with Iowa Institute of Hydraulic Research) | |
| University of Michigan | | |
| Bram vanLeer | <i>Calculation of Hypersonic Flows with Strong Surface Blowing</i> | 176 |
| University of Minnesota | | |
| Donald G. Truhlar | <i>Scattering Theory and Calculations for Chemical Reactions and Molecular Energy Transfer.....</i> | 174 |
| U.S. Army Aeroflightdynamics Directorate—AVSCOM | | |
| W. J. McCroskey | <i>Aerodynamic Flows about High-Performance Rotor Blade Tips.....</i> | 105 |
| | (with NASA Ames Research Center) | |
| W. J. McCroskey | <i>Airloads and Acoustics of Rotorcraft.....</i> | 106 |
| | (with NASA Ames Research Center) | |
| Roger C. Strawn | <i>Computational Fluid Dynamics Prediction of Advanced Rotor Performance</i> | 159 |
| | (with NASA Ames Research Center) | |
| U.S. Army Missile Command—Redstone Arsenal | | |
| Bill J. Walker | <i>Tactical Missile Aero-Propulsion Interaction</i> | 178 |
| ViGYAN, Inc. | | |
| Peter M. Hartwich | <i>General, Fast, Accurate Floating Shock Fitting Procedure for Unadapted Meshes</i> | 62 |
| | (with NASA Langley Research Center) | |
| Peter M. Hartwich | <i>Navier-Stokes Solutions for Vortical Flows over a Forebody</i> | 63 |
| | (with NASA Langley Research Center) | |
| C.-H. Hsu | <i>Prediction of Vortical Flows Using Incompressible Navier-Stokes Equations.....</i> | 72 |
| | (with NASA Langley Research Center) | |
| D. A. Naik | <i>Transonic Potential Flow about Transport Aircraft.....</i> | 121 |
| | (with NASA Langley Research Center, The Boeing Company) | |
| N. J. Yu | <i>Development of an Euler-Based Method for Turboprop Integration</i> | 191 |
| | (with The Boeing Company, NASA Langley Research Center) | |
| Virginia Polytechnic Institute and State University | | |
| Robert W. Walters | <i>Development of Unstructured and Nonequilibrium Chemistry Upwind Algorithms for Hypersonic Flows</i> | 179 |
| Vista Research, Inc. | | |
| Charles L. Rino | <i>Scattering from Ocean Surfaces and Near-Surface Objects.....</i> | 143 |
| Wright Research and Development Center | | |
| Gary W. Huband | <i>Numerical Simulation of an F-16A at Angle of Attack</i> | 74 |

The Navier-Stokes Mach contours
at transonic speeds about a
generic high-speed shape

NASA

National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035-1000

